DOCUMENT RESUME

ED 262 267 CE 042 703

AUTHOR	Sinn, John W.
TITLE	Computer Numerical Control: Instructional Manual. The North Dakota High Technology Mobile Laboratory Project.
INSTITUTION	North Dakota Univ., Grand Forks. Dept. of Industrial Technology.
SPONS AGENCY	North Dakota State Board for Vocational Education, Bismarck.
PUB DATE	85
NOTE	132p.; For related documents, see CE 042 701-705.
PUB TYPE	Guides - Classroom Use - Guides (For Teachers) (052)
EDRS PRICE	MF01/PC06 Plus Postage.
DESCRIPTORS	Competency Based Education; *Computer Oriented Programs; Course Content; Equipment; Industrial Arts; Learning Activities; Lesson Plans; *Machine Tools; *Manufacturing; Microcomputers; *Numerical Control; Postsecondary Education; Secondary Education; Teaching Methods; Vocational Education; Workshops
IDENTIFIERS	*Computer Assisted Manufacturing

ABSTRACT

This instructional manual contains five learning activity packets for use in a workshop on computer numerical control for computer-aided manufacturing. The lessons cover the following topics: introduction to computer-aided manufacturing, understanding the lathe, using the computer, computer numerically controlled part programming, and executing a part program. Each learning activity packet lists prerequisites, purpose, and competencies to be developed and contains information sheets, illustrations, and learning activities. A list of directions for operating a computer numerically controlled lathe is included in the manual. (KC)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

ND8513203

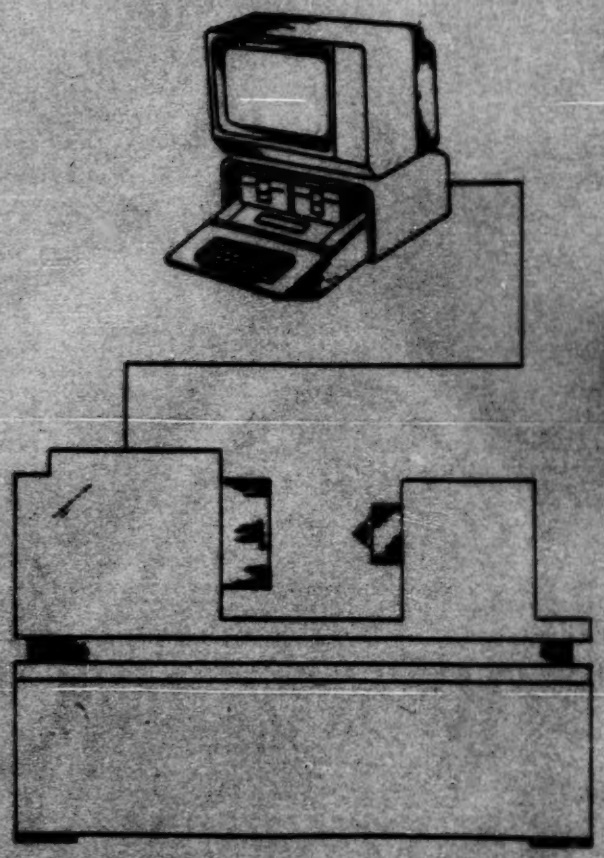
COMPUTER NUMERICAL CONTROL:

Instructional Manual

Writer: Dr. John W. Sinn

ED262267

BEST COPY AVAILABLE



U.S. DEPARTMENT OF EDUCATION
NATIONAL INSTITUTE OF EDUCATION
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)

☑ This document has been reproduced as received from the person or organization originating it.

☐ Minor changes have been made to improve reproduction quality.

• Points of view or opinions stated in this document do not necessarily represent official NIE position or policy.

"PERMISSION TO REPRODUCE THIS MATERIAL HAS BEEN GRANTED BY

L Barnhart

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

CE042703

The Industrial Technology Department, University of North Dakota --
High Technology Mobile Laboratory Project Funded by the North Dakota State Board
For Vocational Education -- Mr. Jerry Bohlman and Dr. Myron Sander, Co-Directors

1985

The North Dakota
High Technology Mobile Laboratory Project

Conducted Through
The Department of Industrial Technology
University of North Dakota
Grand Forks 58202

Funded By
The North Dakota State Board For
Vocational Education
State Capitol
Bismarck, North Dakota

Project Co-Directors:
*Dr. Myron Bender
Department of Industrial Technology
*Mr. Jerry Balistreri
Former North Dakota State
Industrial Arts Supervisor

Specialty Consultant & Writers:
*Dr. Herbert J. Auer
Robotics Instructional Manual
Photovoltaic Instructional Manual
*Dr. Wan-Lee Cheng
Computer-Aided Design
Instructional Manual
*Dr. John W. Sinn
Computer Numerical Control Lathe
Instructional Manual
*Dr. Luvern R. Eickhoff
Laser and Fiber Optics
Instructional Manual

FORWARD

The following five learning activity packages (LAPs) were developed for use with the Paxton-Patterson CNC lathe package, as part of the North Dakota State Board for Vocational Education Mobile High Technology Laboratory. The LAPs can be used seperately although you may find it advantageous to consider the following points, depending on your teaching situation.

LAP I, Introduction to Computer Aided Manufacturing (CAM) should probably be completed by all students, since most students will not have a very good grasp of the overall picture. LAP I is designed to give students an overall picture of computer aided manufacturing. LAP II, Understanding the Lathe, may not be necessary if your students have some prior experience on the metal cutting lathe. LAP III, Using the Computer, also may not be necessary for all students, depending on their past experiences with computers. Clearly, however, this is a useful/necessary LAP for helping to use the computer. LAPs IV and V, CNC Part Programming, and Executing a Part Program are both necessary and should probably be used in their logical order.

The activities in the Paxton-Patterson CNC training blue folder are designed to be worked through logically, as identified in LAPs IV and V. After completion of the LAPs and the first two activities from the blue notebook, you should encourage your students to complete as many activities as time will permit.

On the practical side, a sufficient quantity of 2" brass, aluminum, or plastic is also included if you would like to challenge your students to adjust the programs to different size materials, although this should be done only after a thorough understanding of standard sized, 2" x .500", work is gained.

A duplicate copy of the blue and orange notebooks is included in the Mobile Lab, and also Dr. Myron Bender at UND has a copy of each if the originals should happen to be misplaced. Please feel free to call UND, Industrial Technology Department at 701-777-2249 if you encounter problems with the Paxton-Patterson system. Also, Mr. Ken Kerth, Paxton/Patterson can be reached by calling 1-800-323-8484.

OPERATING INSTRUCTIONS;
Paxton/Patterson CNC Lathe

1. Unlock and open cabinet.
2. Remove wing nuts and move lathe forward on bench.
3. Remove foam rubber packing from around computer, power supply, and cassette tape player. Move computer and tape player to forward right of bench to prevent chips from lathe reaching them.
4. Be certain power strip switch is OFF. Then plug ROM cartridge into right side of computer.
5. Plug lathe motor power cord into wall outlet on left side of van, NOT into power strip.
6. Turn power strip switch ON.
7. Power other things in the following order:
   a) Turn DC power supply ON.
   b) Turn monitor ON.
   c) Turn computer ON.

   (A three line message, followed by OK, should appear on the screen.)

8. Load Master Program into computer via tape player:
   a) Be certain tape is rewound! If not, rewind.
   b) Press Play button.
   c) Type CLOADM and press ENTER on computer.
   d) Flashing F will appear on screen, meaning Found.
   e) OK will appear on screen, meaning loading is complete and tape player will stop.
   f) Press stop on tape player and rewind tape.
9. Load desired PART Program into computer via tape player:
   a) Be certain tape is rewound. If not, rewind.
   b) Press play button.
   c) Type CLOAD Exercise # desired and press ENTER on computer. Not that # can be any of those listed on the tape cassette.
   d) Computer will search for the # specified and a flashing F will appear when it is Found.
   e) OK will appear on screen and tape player will stop.
   f) Press stop on tape player.
10. Type RUN and press ENTER.
11. SETUP will appear on screen. Press ENTER.
12. PART DIA. = will appear on screen. Type 0.5 and press ENTER.
13. PART LENGTH = will appear on screen. Type 1.0 and press ENTER.
14. STOCK DIA = will appear on screen. Press ENTER. This will cause only the graphics to appear on screen.

15. While the drawing is shown on screen, prepare the lathe and work-piece by:

    a) Placing the workpiece in the chuck with 1-1/8" protruding beyond the chuck jaws.
    b) Adjusting the cross feed and longitudinal feed so tool just touches corner of workpiece, then back off both feeds 0.050" (one turn).
    c) Turning the lathe motor ON and adjusting the control to about mid-speed position.
    d) Pressing ENTER again to remove drawing from screen.
    e) NEW SCALE- Y OR N? will appear on screen.
    f) Type N and press ENTER.

16. STOCK DIA. = will appear on screen. Type 0.5 and press ENTER. The stepper motor(s) will begin operation.

17. When it has completed the cut(s) specified in the loaded program the stepper motor(s) will have stopped in their original positions.

18. If another workpiece with the same cut(s) is to be run, press ENTER.

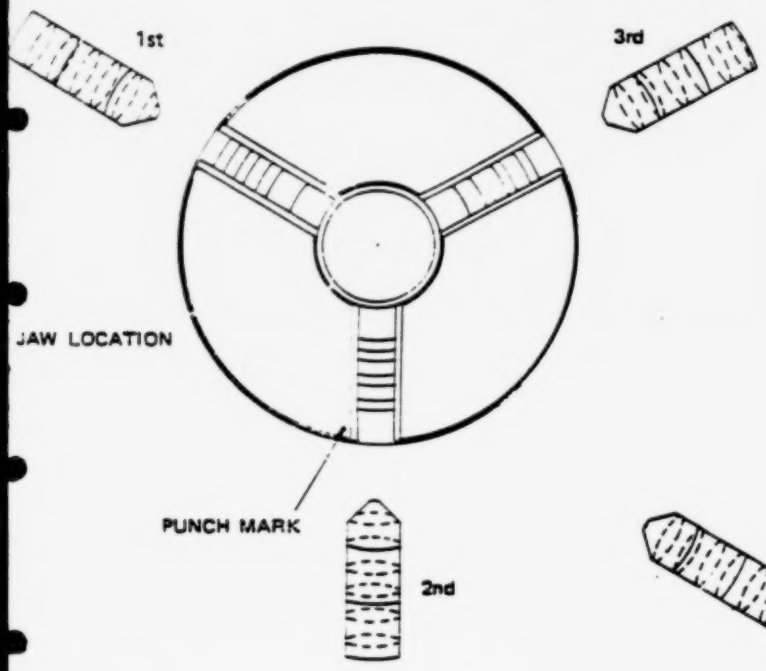
    a) NEW SCALE- Y OR N? will appear on the screen.
    b) Type N and press ENTER.
    c) Insert workpiece as per steps 15a through 15d above.
    d) STOCK DIA. = will appear on screen. Type 0.5 and press ENTER.

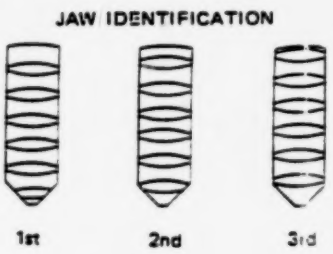
19. If a different Program is to be used:

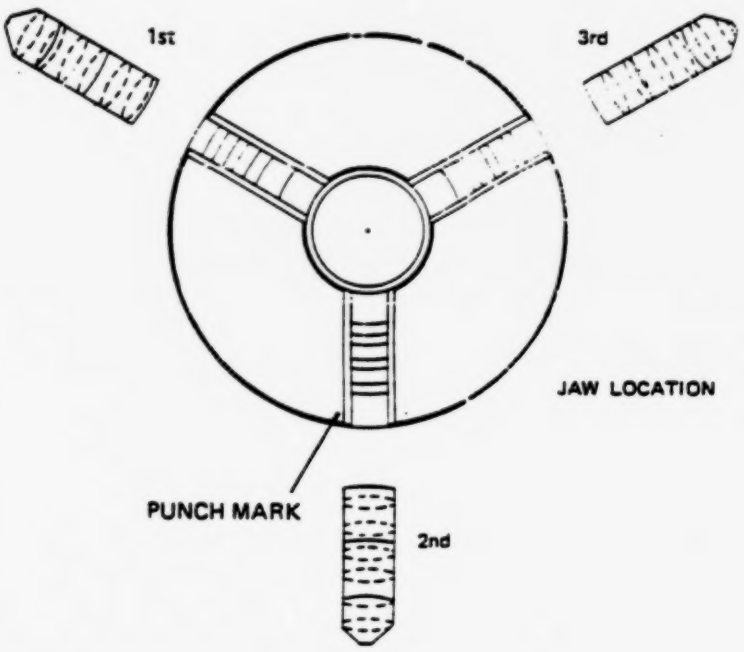
    a) Rewind the tape only if the # is lower than the # just used.
    b) Press ENTER then BREAK.
    c) Load a new Program into the computer via procedures of steps 9b through 9f above.

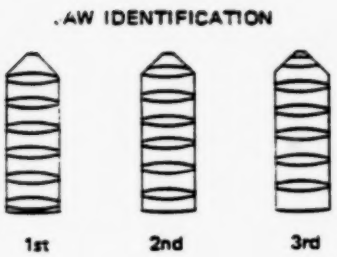
THREE JAW CHUCK

The three jaw self-centering chuck is the most popular of all lathe attachments. This chuck will grip round or hexagonal work quickly since the jaws move simultaneously to automatically center the work being held. The jaws on the chuck are designed so that the same chuck can be used for both internal and external gripping. Jaws are reversible for holding larger diameter work. Due to the nature of the design of a three jaw chuck, it cannot be expected to run perfectly true. Even three jaw chucks costing 5 times as much as the one made for this lathe will have a 0.002" to 0.003" runout. If perfect accuracy is desired in a particular operation, the use of a four jaw chuck or collets are recommended.









IMPORTANT

The three jaw chuck is designed to take up to 1-3/16" diameter stock with the jaws in the normal position. For larger diameter work, reverse the jaws. To prevent permanent damage, finished, turned, or drawn stock should only be held with this chuck. For rough castings, etc., use the four jaw chuck. Do not overtighten the chuck. Use only moderate pressure with the Tommy bars supplied.

To reverse the chuck jaws, rotate the knurled scroll until the jaws can be removed from the chuck body. After the jaws are removed, they can be easily identified by the location of the teeth to the end of the jaw (see drawing). To maintain chuck accuracy, the 2nd jaw must always be inserted in the same slot even when the jaws are reversed. This slot is identified by a punch mark next to the slot. (See drawing). Always insert the jaws in the order and location as shown on the drawings. Turn the scroll counterclockwise, when viewed from the face of the chuck until the outside start of the scroll thread is just ready to pass the slot for the 1st jaw. Slide the 1st jaw as far as possible into the slot. Turn the scroll until the 1st jaw is engaged. Due to the close tolerances between the slot and jaw, the most difficult part of replacing the jaws is engaging the scroll thread and 1st jaw tooth without binding. Therefore, never use force when replacing the jaws and if binding occurs, back up the scroll slightly and wiggle the jaw until it is free to move in the slot. Advance the scroll and repeat for the 2nd and 3rd jaws. The scroll thread must engage the first tooth in the 1st, 2nd, and 3rd jaws in order.

BEST COPY AVAILABLE

COMPUTER AIDED MANUFACTURING

TABLE OF CONTENTS

Forward	page ii
LAP	Number
Introduction to Computer Aided Manufacturing	I
Understanding the Lathe	II
Using the Computer	III
CNC Part Programming	IV
Executing a Part Program	V

UNIVERSITY OF
NORTH DAKOTA
DEPARTMENT OF INDUSTRIAL TECHNOLOGY

HIGH TECHNOLOGY MOBILE LABORATORY

Learning Activity Packet

MAJOR INSTRUCTIONAL UNIT: COMPUTER AIDED MANUFACTURING

LEARNING ACTIVITY PACKET (LAP) NUMBER: I

LAP TITLE: Introduction to Computer Aided Manufacturing

PREREQUISITE LAP (S) : None

LAP PURPOSE: This lap serves to introduce the student to computer aided manufacturing.

COMPETENCIES TO BE DEVELOPED:

After completion of this unit the student should be able to identify the major applications of computer aided manufacturing in industry.

LEARNING STEPS:

INTRODUCTION TO COMPUTER AIDED MANUFACTURING

1. READ - Unit objective and specific objectives, pages 1 - 3.
2. STUDY - Background information, pages 4 - 17.
3. DO - Learning Activities, pages 18 - 19.
4. STOP - See your instructor for evaluation. If your work is satisfactory, go to step 5. If not, go to step 2.
5. REVIEW - Objectives, background information, learning activities.
6. TAKE - The LAP Evaluation, pages 22 - 25.
7. STOP - Give your instructor the completed LAP evaluation. If you scored 85 percent or more go on to lab II. If not, go to step 5.

INTRODUCTION TO COMPUTER AIDED MANUFACTURING
UNIT I

UNIT OBJECTIVE

After completion of this unit, the student should be able to identify the major applications of computer aided manufacturing in industry. This knowledge will be evidenced by correctly performing the procedures outlined in the learning activities and by scoring at least 85 percent on the unit test.

SPECIFIC OBJECTIVES

After completing this unit, the student should be able to:

1. Define three computer analysis and control applications used in manufacturing.
2. Define four decision making areas common to manufacturing operations analysis concerns.
3. Define computer aided design as related to computer aided manufacturing.
4. Define numerical control as related to computer aided manufacturing.
5. List three characteristics which should be exhibited by manufacturing processes if they are to be run by numerical control.
6. Define the three major components in a numerical control system.
7. List the six steps involved in the typical numerical control function.
8. List nine advantages and disadvantages of numerical control.
9. Define the difference between numerical control and computerized numerical control.
10. List four specific computerized numerical control functions in manufacturing.
11. List six applications for a robot in manufacturing.
12. List six considerations to be aware of when selecting robotic applications in manufacturing.
13. Define whatis:meant by an automated factory, citing an example of how this might work.
14. Define five "people related" concerns involved with comptuer aided manufacturing.

Introduction To Computer Aided Manufacturing

Unit I

Background Information

Computers and automation have become one of the realities of the technological age. Computer and automation applications and uses are virtually endless with their proliferation into manufacturing becoming increasingly pronounced on an almost daily basis. This chapter is an attempt to briefly overview the current major manufacturing applications of the computer, including computer analysis and control, computer aided design, numerical control, computer aided manufacturing, computerized numerical control, robotics and computerized interface for automated factories.

Computer Analysis and Control

One example of computer control is scheduling and routing. Scheduling and routing is aided by the computer by achieving an efficient balance of major inputs and desired outputs of manufacturing. As with many currently computerized functions, the scheduling and routing was at one time done with pencil and paper or in the manager's head. With the advent of the computer, the more complicated manufacturing setting involving perhaps thousands of machines, workers, materials, and several products can now accurately and efficiently be analyzed to provide optimum job-lot sizes, machines, numbers of workers and other key manufacturing inputs. Computer programs are written indicating maximum and minimum limits for materials to be used in manufacturing. During manufacturing, as levels of materials needed become apparent, the computer recognizes this and calls up additional quantities. Similarly when planning the routing for manufacturing if all limitations, locations, capabilities, etc., in the manufacturing setting are inputted into the computer, the computer can be used to aid in routing. Various types and sizes of computers can be applied in scheduling and routing, ranging in size from desk top types to large centralized units serving an entire factory. A sample program used

as part of computer analysis and control in inventory is shown in figure 46. In this simple inventory analysis various equations are supplied (Hopeman, pp. 362-377) and written into the program in an appropriate manner allowing for all the necessary inputs relating to desired outputs. This information is typically included as part of the materials requirement plan for scheduling and routing. Not only is this type of scheduling and routing useful for optimizing short-term production by maximizing work processing centers and people, minimizing bottlenecks, decreasing inventory costs, having raw materials scheduled on time, and perhaps in other ways, but perhaps more importantly if the information is stored over time a data base can be provided from which to aid in plant layout, expansion and capital investment projects.

Related to scheduling and routing, cost analysis and control uses the computer for cost accounting purposes to monitor and determine cash flow related to manufacturing. Industrial cost accounting divisions study monetary activities related to processes, materials, products, departments and other general cost factors (Jensen and Wright, p. 183). Information supplied to the cost accounting division will aid in determining budgets, profits, production levels, which products are profitable and should be continued, general efficiency levels, expansion issues, and other manufacturing matters. Often this information will be stored in the computer as a data base for future decisions. Computerized cost accounting activities subsume many specific cost analysis tools, including break-even analysis, capital investment analysis, inventory analysis, and product cost analysis. While many cost activities may not be a direct function of cost accounting, most cost analysis tools are indirectly contingent on cost accounting for information necessary for their decision making input and day-to-day functioning. Related to cost accounting, another computer application which should be studied by manufacturing students is data

```
 10 REM* Level II, Radio Shack, Summer 1982 by John W. Sinn
 20 REM* Inventory Analysis Modified From Henry' S*
 30 CLS : Print "This Program Will Give An Inventory Analysis"
 40 Input "What Are The Annual Requirements (Total No. of sub-
    units needed?); R
 50 Input "What is the Lot Size of Sub-Unit Quantity Per
    order?"; Q
 60 Input "What is the Procurement Cost Per Order?"; S
 70 Input "What is the Total No. of Orders Per Year?"; R/Q
 80 Input "What is the Lead Time Needed Per Order?"; L
 90 Input "What is the Time the Supply will Last?"; T
100 Input "What is the Max. Inventory Level? ; MA
110 Input "What is the Min. Inventory Level?"; MI
120 Input "What is the Carrying Cost Per Unit?"; C
130 APC = (R/Q) * S  OPY = R/D
140 ACC = (Q/2) * C
150 E = APC + ACC
160 EOQ = SQR(2RS/C)
170 U = MA - MI/T
180 RP = (u * L) + MI
190 Print "Annual Procurement Cost: $" ; APC
200 Print "Annual Carrying Cost:  $"; ACC
210 Print "Economic Order Quantity: "; EOQ; "Units"
220 Print "Total Incremental Cost: ?"; E
230 Print "Use Rate: "; U;" Units "
240 Print "Reorder Point: "; RP; " Units"
250 End
```

Figure 46. A Sample Inventory Analysis And Control Program.

collection and processing. For example, in an order processing situation, quantity and types of products ordered are entered into the computer when the order is received. The input is matched against the inventory and the inventory record is reduced. The computer prepares a shipping order, invoice record and shipment schedule, based on quantity. Other computer data collection and processing applications in manufacturing could include salary and wage payments, income tax withholdings, payroll deductions, among others. Manufacturing operations analysis is another computer application area which should be studied. The following decision making areas indicate common manufacturing operations analysis concerns for computer applications:

1. If a product can be made at more than one plant location, how much manufacturing should occur at each plant?

2. If products can be shipped from more than one plant or warehouse, which plant or warehouse should service which customers?
3. When and how often should materials be ordered from suppliers to keep inventory levels at a minimum without creating production line shortages of materials?
4. To meet finished goods manufacturing requirements, how many machine-hours should be scheduled for next week/month? How many labor hours? How many shifts? Is overtime necessary?

Computer Aided Design

Computer aided design permits a designer to work at the screen of an interactive graphics terminal to develop a geometric model which describes the size and shape of a component to be produced. This model establishes a geometric data base that may be used to structurally analyze the component, to test various features such as fit and tolerance of the component, or to produce working drawings of the component. The shapes generated as part of the model can be edited to conform to desired specifications. Many of these systems now have processing limitations programmed into the computer so that when the programmer inputs information beyond the capability of the processes which will be used to produce the component, the inputted information is rejected. When this occurs it signals the designer to go back and start over, or adjust whatever inputted request was beyond the processing capability. One of the predominant problems with computer aided design is that it is rather simple and easy to generate what appears to be a workable design when sitting at the CRT terminal. However, when this computer aided design is actually placed in the manufacturing environment with the numerous unpredictable variables inherent, the CAD approach often falls short. When this is coupled with programmers who are relatively inexperienced in both programming and manufacturing, the problems of CAD become even more compounded.

Numerical Control

Numerical control (NC) can be defined as form of programable automation in which the process is controlled by numbers, letters or symbols. In NC, the numbers form a program of instructions designed for a particular workpart job. When the job changes, the program of instructions is changed. This capability to change the program for each new job is what gives NC its flexibility. It is much easier to write new programs than to make major changes in manufacturing equipment. NC equipment is used in all areas of manufacturing, although its primary application is related to metal. Today approximately 15% of all machines in manufacturing are NC, including drills, mills, lathes, grinders, presses, welders, among others. NC should be considered for manufacturing operations possessing the following characteristics:

1. Workparts produced in various sizes;
2. Workparts produced in small to medium-sized batches; and,
3. A sequence of similar processing steps is required to complete the workpiece.

NC systems consist of a program of instructions, a controller unit, and a machine tool. The program of instructions is the detailed step-by-step set of instructions which tell the machine what to do. It is coded in numerical or symbolic form on some type of input medium which will be interpreted by the controller unit. The most common type of NC program and tape is shown in figure 47. As can be noted on the program, actual sizes and distances are indicated in their real size. The second basic component of the NC system is the controller unit. This consists of the electromechanical hardware that reads and interprets the program of instructions and converts it into mechanical actions/motions at the machine. The third basic component of the NC system is the actual machine which performs useful work. The process for use of NC in manufacturing commonly occurs as follows:

```
%
N1G90G80G40G0XZ.Y-3.8295M0
N2G46Z.9H7S3400M3
N3G81Z-.38.IF3000
N4Y-5.8295
N5X-2.
N6Y-3.8295
N7X-6.25
N8Y-5.8295
N9X-10.25
N10Y-3.8295
N11X-14.5
N12Y-5.8295
N13X-18.5
N14Y-3.8295
N15X-22.75
N16Y-5.8295
N17X-26.75
N18Y-3.8295
N23G80
N24X-27.031Y-1.698
N240M20
N241M20
/N25Z.1
/N26G99G81Z-2.61R-2.35
/N33X-27.625Y0
/N34X-27.031Y1.698
/N35X-22.469
/N36X-21.875Y0
/N37X-22.469Y-1.696
/N38X-18.781
/N39X-19.375Y0
/N40X-18.781Y1.698
/N41X-14.219
/N42X-13.625Y0
/N43X-14.219Y-1.698
/N44X-10.531
/N45X-11.125Y0
/N46X-10.531Y1.698
/N47X-5.969
/N48X-5.375Y0
/N49X-5.969Y-1.698
/N50X-2.281
/N51X-2.875Y0
/N52X-2.281Y1.698
/N53X2.281
/N54X2.875Y0
/N55G98X2.281Y-1.698
/N56G80M9
/N57Z5.
```

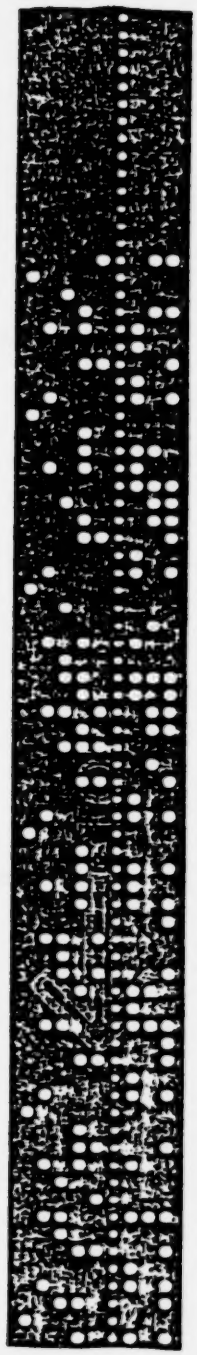


Figure 47. CNC Program And Tape.

BEST COPY AVAILABLE

1. The working drawing of the workpart must be interpreted in terms of processes to be used;
2. Actual dimensions and specifications are taken from the drawing and handwritten into a program;
3. The programmer types the program from the initial hand-written program;
4. A punched tape (perhaps many feet long) is prepared by hand from the typed program;
5. The tape is checked/verified by running it back through the printer and receiving a program printout which is checked for any errors; and,
6. The job is produced on an NC machine under tape control and modified if needed prior to use in manufacturing.

Some of the advantages and disadvantages of NC include:

1. Relatively easy design changes. Holes can be added, moved, or omitted simply by changing the tape;
2. Simplified quality control since the tape will produce the same accuracies part after part as long as the cutters and machinery are maintained;
3. Reduced scrap as a result of better quality control;
4. Reduced space requirements as a result of the tape being smaller as compared to jigs/fixtures which were often used formerly;
5. NC equipment is generally three to six times the cost of conventional manual equipment. However, many processes are often put in what are called "machining centers" with one new NC machine replacing several former processes, thus making NC an attractive investment;
6. New people must be trained, including electronic and mechanical service persons, programmers and machine operators. However, most existing workers can be "re-trained" if they are willing. Also, it should be noted that with the advent of NC the operators role

is less difficult, often requiring them only to put a workpiece on the machine table and start the machine (tape). However, it is also true that now the NC operator will often be responsible for more than one machine or an entire "machining center". With NC the operator is nearly eliminated in the conventional sense of "Running" a machine. This explains, in part, why workers appear to be idle in between machining parts which are being produced - the NC is literally running the machine process;

7. Extremely careful planning of cutting tools and processes, usually coordinated with drafting, is necessary if small details are to be built into the component;
8. NC machines do not cut any faster than conventional tools. metal (and other) materials can only be removed at a given rate and cutters will prematurely be destroyed if speeds and feeds are excessive. However, NC processes are more consistent since they do not fatigue, have family or human relations problems, and they do not come in late for work; and,
9. People will often resist changes such as NC since they see this type of change as a threat to their "job security". The fact is that people, generally, will not be put out of work, but rather, as pointed out above, they will need to retrain as a programmer or technician, however, they will need to change.

Computer Aided Manufacturing/Computerized Numerical Control

Computer aided manufacturing (CAM) consists of using the computer to control various factors related to processing components being produced. Piggybacking on what was previously stated about CAD, many (if not all) of the inputs generated as part of the CAD data base typically can be used to produce a numerical control tape which can be cycled via the computer

to control processing centers to produce the component generated earlier in CAD. Identified as generative programming, this CAD-CAM relationship is increasingly being used in industry to identify material to be machined, select tooling to be used; to aid in determining fabrication steps, and to sequence the tool paths. Computer aided manufacturing typically uses machining centers consisting of a single (sometimes two) machine tools equipped with automatic tool changers and capable of performing a wide range of operations on a common workpiece. Typical operations performed at a CAM machining center might include milling, facing, slotting, boring, drilling and tapping on the same workpiece. Often the computer controls the tooling which is "called-up", the sequencing of operations, durations of cuts, application of cutting fluid, sensing tool wear, measuring for quality and so on. Direct savings are provided to the manufacturer with CAM because of the increased productivity and reduced in-process inventory costs (Yankee, p. 278).
The CAM operation is sometimes referred to as computerized numerical control (CNC). As mentioned in an earlier section of this chapter, NC tapes are created for purposes of performing various operations. Often the NC tape is generated as a direct function of design as indicated above. When the tape is completed it can be controlled via the computer at a single processing or machining center with a single tool (which can be manually changed by the operator) or with a multiple tool machine as described earlier in this section. CNC is different from CAM since the primary function of the computer is CNC is related to the NC tape, whereas in CAM the computer is often used for various manufacturing analysis functions such as sequencing operations, planning tool paths and determining fabrication steps, as indicated above. CNC is a more limited computer application in manufacturing, generally using a microprocessor as part of the machining center as shown in figure 48. Specific CNC functions in manufacturing could include:

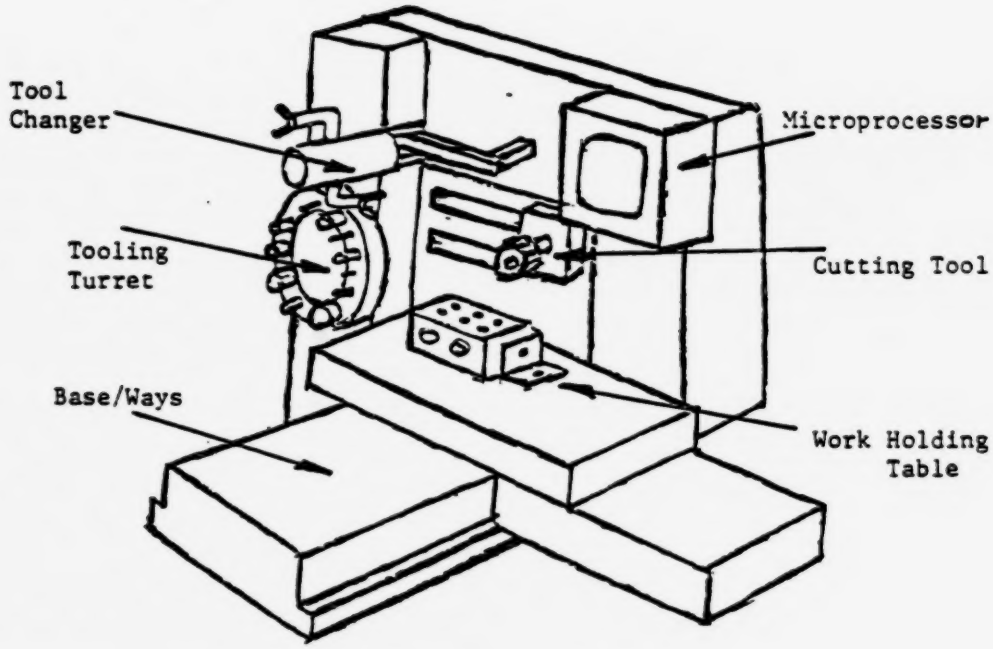


Figure 48. CNC Machining Center.

1. Storing NC tape information in the computer, thus not requiring the mechanical tape after stored;
2. Once stored, corrections can be inputted into the computer. After computer corrections are made, the computer can generate a new tape, providing the corrected tape for permanent storage;
3. Indicating errors in the NC tape, later to be manually corrected;
4. Controlling the start and stop of the actual operation, aside from loading/unloading workpieces/fixture and pressing the button to start the process/tape.

Robotics

Robots are operatorless devices which are programable. The largest users of industrial robots in the U.S. today are automotive manufacturers. Japan has most of the robots which are in operation worldwide, and they are a major

manufacturer of robots. In some situations robots have demonstrated that they can handle heavier loads more accurately at higher speeds than humans. The need to increase productivity and reduce cost in manufacturing has contributed to their increased application in recent years. Robots are available with handling capacities that range from minisized parts of a few ounces and a reach of a few inches to huge robots capable of payloads up to seven hundred pounds. Robots can be programmed similar to other automated manufacturing machines discussed earlier in this chapter. Some robots are produced with sophisticated memory systems having the capability of being able to take alternative actions under changing conditions, while other systems are in use with the same overall utility, but with a limited work-handling flexibility. These are generally used for routine jobs involving single "put-and-take" motions. It is possible to "teach" some robots to do a specified job by switching its control system to a recording mode and manually moving the robot arm through subsequent operations to be performed. As can be seen in figure 49 the robot has a

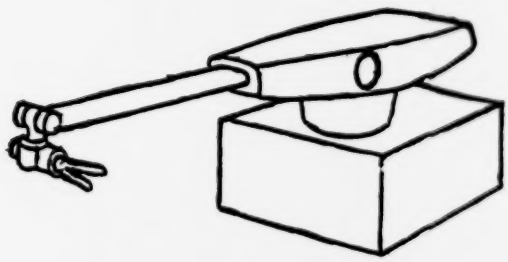


Figure 49. The Industrial Robot.

mechanical arm which can be moved in a variety of directions. Also a variety of hand-like fingers or grippers, vacuum cups, or a tool such as a welding gun or impact wrench can be attached and operated by the robot. Robotic capabilities have become common place in many manufacturing industries. Among the typical jobs performed by robots are assembly operations, materials movement, die casting, forging, stamping presses, welding, plastic injection and compression

molding, investment casting, spray coating/painting and heat treatment operations. There are several important factors to consider when selecting robotic application. Among these considerations are:

1. Speed of movement;
2. Workpiece weight;
3. Accuracy and agility of workpiece positioning;
4. Difficulties of handling a specific type of workpiece (shape, uniformily, etc.);
5. Type of layout (continuous/intermittent);
6. Cost of labor relative to cost of the automated robotic operation; and,
7. Need for human as opposed to mechanical inputs in the manufacturing operation.

Robots are helping to eliminate the risks involved to workers in some hazardous operations such as metal working presses, hot metal handling, chemicals handling among others. Also injuries can often be prevented since lifting/positioning heavy objects can be eliminated. Robots are ideally suited to parts-handling and assembly operations in hostile or unpleasant manufacturing environments.

Computer Interface For Automated Factories

At a basic level CAD-CAM interfaced provides a sophisticated system for combining design and manufacturing functions to increase efficiency. Essentially CAD-CAM systems use CAD generated data bases to perform CAM operations, thus gaining design and manufacturing functions with a single effort. In fact, many manufacturing operations are increasingly relying on CAD data to be used to program their numerical control tapes used for various shop floor operations. When design, manufacture and control funtions are combined it can approach being a computerized factory. Theoretically, an automated factory could be operated by an individual at a graphic terminal, developing a component design, feeding the design into N.C. equipment and

monitoring and controlling with various scheduling, routing, inventory and other programs. While it may be possible to have computerized peopleless factories in the future, currently people are very much a part of the manufacturing process.

When many systems are computerized in the continuous production setting consistent work speeds, tool wear, materials flow, tooling changing time, quality levels, and other factors can result in higher productivity. This is because the computerized factory system has tighter control throughout, providing more predicatable production circumstances and results. Many systems throughout the world are currently using computerized robots and materials handling systems as part of the computerized factory (Lindberg, pages 562-573). In these systems master control schedules are generated based on known production capabilities in the factory. The master schedule will then control materials and/or components called out of inventory stocks and their routing to various appropriate machining or assembly centers. Often the load/unload function is computerized from the inventory point completely through the point of finished product, completing the computerized factory.

Final Points Relating to Automation

1. Automation is Expensive - Automation in virtually any form is capital intensive and will usually require significant capital outlays. Consequently it takes large volumes/lot sizes of production to make most forms of automation cost effective. However, with good planning, properly selected automated equipment can provide productivity increases needed by industry.
2. Automation and New Jobs - Automation generally means new jobs will be created to service the new types of equipment. Programmers will be needed, electronic technicans, computer managers, and so on, all

are new types of workers in tommorrow's production setting. However, it is also reasonable to conclude that some of the old jobs will be lost due to the automation. As was shown earlier in this chapter, often the older machines are grouped into one larger processing or machining center which is fully automated. In the process, several traditional positions may have been eliminated, however the traditional positions' worker can now be retrained to become involved with the automation.

3. Automation Means Working Smarter - The automated factories of tommorrow require "brain intensive" workers. Clearly, it will require more than just a high school diploma to be able to even work on the assembly line. Workers must be well rounded in both traditional production content as well as math and science, among other areas. It is quiet likely that the automated factories will require standard B.S. degreed people for entry-level positions in the future.

4. Skilled or Unskilled? Clearly, the gap between skilled and unskilled people is widened by automation. Individuals without the skills or knowledge, and perhaps without the ability/willingness to learn will relegated to the unskilled tasks in the future - all of which are becoming in shorter supply.

5. Automation Supervision - Automation raises the skill level of the employee to that of supervisory level. This is particularly true since the automation worker will quite likely be responsible for some extremely expensive equipment, as well as for reasons already mentioned.

6. Worker and Management - The automated factory requires more, better trained workers. The better trained, highly skilled prople will demand the respect of management since the equipment cannot function without the skilled people.

INTRODUCTION TO COMPUTER AIDED MANUFACTURING

UNIT I

LEARNING ACTIVITIES

DIRECTIONS

The following activities are provided to allow you to reinforce your learning of Unit I. Complete each activity exactly as prescribed and do not proceed to Unit II until all activities for unit I have been accomplished.

1. Look up each of the following words and phrases and then define them in your own words:

   Computer Analysis and Control _____

   _____

   Computer Aided Design _____

   _____

   Numerical Control _____

   _____

   Computerized Numerical Control _____

   _____

   Robotics _____

   _____

   Computer Interface For Automated Factories _____

   _____

2. Explain why you think the computer will probably be used increasingly in manufacturing _____

   _____

   _____

3. Explain what you think will be the role of people, related to computers in manufacturing, in the future. _____

   _____

   _____

4. Computer aided equipment in manufacturing is among some of the most expensive equipment in industrial activity. If the equipment is so expensive, why are manufacturers quite willing to purchase the CAM equipment. Please explain why this is true. _____

_____

_____

_____

5. Explain (a) the function of the tape in both NC and CNC manufacturing operations, and (b), why the tape is not necessary in CNC operations

_____

_____

_____

_____

_____

6. Explain why some people are concerned about the possibility of losing their jobs to CAM equipment._____

_____

_____

_____

INTRODUCTION TO COMPUTER AIDED MANUFACTURING

UNIT I

EXPECTED RESULTS OF LEARNING ACTIVITIES

1. Computer Analysis and Control - Using the computer to analyze and control a variety of manufacturing activities, including inventory, materials flow, line balancing, personnel requirements, scheduling and routing, cost analysis, and so on.

   Computer Aided Design - Designing products/components by working at the screen of an interactive graphics terminal to develop a geometric model, including size and shape, generate data base, specifications, and so on.

   Numerical Control - Forms of programmable automation in which the process is controlled by numbers, letters or symbols in the form of a tape program of instructions designed for a particular workpart or job.

   Computer Numerical Control - Using the computer to store and/or modify the NC tape information which would otherwise be useful only in tape form.

   Robotics - Operatorless devices which are programmable, generally through the use of the computer.

   Computer Interface for Automated Factories - Combining CAD - CAM and other computer functions to literally computerized the entire factory. Virtually all activities have the potential of being interfaced with the computer in the automated factory.

2. The computer will be used increasingly in manufacturing because (A) Computers can be more reliable than people; (B) Computer equipment can increase productivity on the shop floor; (C) computer equipment can produce consistent quality levels on mundane tasks all day long with no complaints; (D) computer equipment can perform unsafe tasks; (E) Computers equipment can pay for itself in rather brief periods, on the order of 1-4 years, among other reasons.

3. People will be involved in programming, servicing, designing and fabricating, trying to improve on existing computer systems, and so, clearly increasingly involved with them in the future.

4. Even though CAM equipment can easily cost upwards of one-quarter million dollars, it is true that it often pays back in a relatively brief time period, for a variety of reasons, as already presented. Fundamentally, however, CAM equipment has a tremendous potential for being cost effective.

5. The NC/CNC tape has a series of holes punched in a sequence which can be decoded by the tape reader and encoded into the drive mechanisms of the CAM equipment to provide motion at the manufacturing machine. The CNC has the ability to store/modify information in the computer while NC does not have this ability.

6. Since computers can be interfaced with virtually any mechanical/machine or production activity, it is true that people can be "displaced". The displacement of course, can lead to some "unskilled" individuals being without a job.

INTRODUCTION TO COMPUTER AIDED MANUFACTURING

UNIT I

LAP EVALUATION

Name _____

A. Listed below are terms, phrases and statements frequently used in computer aided manufacturing. Examine each statement closely and determine whether it is true or false. Indicate your answer on the line preceding each.

1. _____ CNC is a form of computer analysis and control.
2. _____ Manufacturing operations analysis is a form of computer analysis and control.
3. _____ A typical manufacturing operations analysis decision is the type of material to use in manufacturing a product.
4. _____ CAD generated designs can be readily placed into production with little effort.
5. _____ NC and CNC are essentially the same type of computer activity.
6. _____ NC is impractical when manufacturing large lots of the same product.
7. _____ NC requires manual inputs into the tape.
8. _____ Design changes in products can be relatively easy with NC.
9. _____ Quality control is generally good with NC.
10. _____ NC equipment is similar in cost to conventional manufacturing equipment.
11. _____ CAD generated information is generally useful for CNC operations.
12. _____ CNC can function without the mechanical tape.
13. _____ American manufacturers are curently using more robots than any other manufacturers in the world.
14. _____ Robots are limited to being able to move/handle a maximum of 100 pounds.
15. _____ Some robots can be "taught" how to repeat some functions.
16. _____ Robots require no human involvement in manufacturing.
17. _____ Robots are not useful in hazardous situations.
18. _____ Automated factories could conceivable run 24 hours a day, in the dark.
19. _____ Automation will only take jobs, not generate any new ones.
20. _____ Unskilled individuals have an increasing role in the CAM environment.

B. Listed below are questions requiring short answers. Read each question carefully and then fill-in your answers on the blanks provided.

1. Define three computer analysis and control applications used in manufacturing.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

2. Define four decision making areas common to manufacturing operations analysis concerns.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

   d. _____

   _____

3. Define computer aided design as related to computer aided manufacturing.

   _____

   _____

   _____

   _____

4. Define numerical control as related to computer aided manufacturing.

   _____

   _____

   _____

   _____

5. List three characteristics which should be exhibited by manufacturing processes if they are to be run by numerical control.

   a. _____

   b. _____

   c. _____

6. Define the three major components in a numerical control system.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

7. List the six steps involved in the typical numerical control function.

   a. _____

   b. _____

   c. _____

   d. _____

   e. _____

   f. _____

8. List nine advantages and/or disadvantages of numerical control.

   a. _____

   b. _____

   c. _____

   d. _____

   e. _____

   f. _____

   g. _____

   h. _____

   i. _____

10. List four specific computerized numerical control functions in manufacturing.

    a. _____

    _____

    b. _____

    _____

    c. _____

    _____

    d. _____

    _____

11. List six applications for a robot in manufacturing.

a. _____

b. _____

c. _____

d. _____

e. _____

f. _____

12. List six considerations to be aware of when selecting robotic applications in manufacturing.

a. _____

_____

b. _____

_____

c. _____

_____

d. _____

_____

e. _____

_____

f. _____

_____

13. Define what is meant by an automated factory, citing an example of how this might work.

_____

_____

_____

_____

_____

14. Define five "people related" concerns involved with computer aided manufacturing.

a. _____

_____

B. _____

_____

c. _____

_____

d. _____

e. _____

INTRODUCTION TO COMPUTER AIDED MANUFACTURING
UNIT 1

LAP EVALUATION KEY

A.
1. False
2. True
3. False
4. False
5. False
6. False
7. True
8. True
9. True
10. False
11. True
12. True
13. False
14. False
15. True
16. False
17. False
18. True
19. False
20. False

B. 1.
a. Scheduling and routing - the computer is programmed to calculate appropriate responses to typical schedule and route problems.

b. Inventory Control - Programs can be written to indicate/analyze minimum and maximum levels.

c. Personnel Allocation - Appropriate numbers of people can be calculated for various functions in the manufacturing operation.

2.
a. If a product can be made at more than one location, how much production should occur at each location?

b. If products can be shipped from more than one plant or warehouse, which plant or warehouse should be used?

c. When should material orders be filled to keep adequate inventories?

d. How many machine/work hours should be scheduled for upcoming periods?

3. Computer aided design is concerned with graphical and analytical design questions at the outset of the product's life while computer aided manufacturing is concerned with computer applications in the actual production or manufacture of the product.

4. Numerical control is a symbolic form of control of machine tools through the use of numbers and letters (or others) to make a tape which can be used repeatedly to control the manufacture of pieces/parts/products. It is related to CAM by virtue of both (CAM and NC) using symbols in a logical manner to gain control.

5.
a. Workparts produced in various sizes.

b. Workparts produced in small to medium sized batches.

c. Sequential processing steps should be used - repeatable.

6. a. Program of instructions - numerical symbols in code form.
   b. Controller unit - This is also called an interface, but in either case it converts the numerical code to useful mehcanical motion.
   c. Machinery/Equipment - The actual machine tool which is being controlled.

7. a. Interpret the working drawing.
   b. Hand write the dimensions of the drawing into a program.
   c. Hand written program is typed.
   d. A punched tape is prepared from the hand typed version.
   e. Verify the tape by running it through a printer.
   f. Produce the part by NC tape application.

8. a. Relatively easy design changes.
   b. Simplified quality control.
   c. Reduced scrap.
   d. Reduced space requirements.
   e. Increased cost for equipment.
   f. Increased training/skills requirements
   g. Careful manufacturing planning is necessary.
   h. Cutting speeds/feeds remain the same.
   i. Change over to NC will probably be resisted by some.

9. Numerical control must use the hand generated tape while computerized numerical control can rely upon the computer for storage and modification functions.

10. a. Storing - NC tape information .
    b. Correcting/Modifying the program, once stored.
    c. Indicating errors in the program.
    d. Controlling the start and stop of the actual operation.

11. a. Materials movement.
    b. Assembly
    c. Die Casting
    d. Painting
    e. Forging
    f. Stamping
    g. Molding
    h. Welding

12.
a. Speed of movement
b. Workpiece weight
c. Accuracy and agility
d. Shape/uniformity of materials/components to be handled
e. Type of manufacturing layout
f. Cost of labor relative to robot investment
g. Need for human as compared to mechanical inputs

13. An automated factory is a factory which can be literally controlled by the computer, including machinery, decisions, designs, analysis, and so on.

14.
a. Job Displacement - People must be aware that the nature and availability of jobs will change as CAM is inputted.
b. Working Smarter - People must realize that they will, of necessity, have to use the brain more when working with CAM applications.
c. Automation Supervision - People actually become supervisors when they are involved with CAM - not all people like this aspect of their changing work environment.
d. Change - People will be required to change on a continual basis with CAM. Again some people reject this.
e. Worker/Management - The worker and management will become more of a "one and the same" relationship. This can create labor relations concerns for some workers/managers in some industries.

UNIVERSITY OF
NORTH DAKOTA
DEPARTMENT OF INDUSTRIAL TECHNOLOGY

HIGH TECHNOLOGY MOBILE LABORATORY

Learning Activity Packet

MAJOR INSTRUCTIONAL UNIT: COMPUTER AIDED MANUFACTURING

LEARNING ACTIVITY PACKET (LAP) NUMBER: II

LAP TITLE: Understanding the Lathe

PREREQUISITE LAP (S) : None

LAP PURPOSE: The LAP is designed to help the student understand the metal lathe.

COMPETENCIES TO BE DEVELOPED: After completion of this unit, the student should be competent at manually operating the lathe; understanding how it operates; and, understanding how to properly care for the lathe.

LEARNING STEPS:
UNDERSTANDING THE LATHE

1. READ - Unit objective and specific objective, pages 1-3.

2. STUDY - Background information, pages 4 - 17.

3. DO - Learning activities, page 18.

4. STOP - See your instructor for evaluation. If your work is satisfactory, go to step 5. If not, go to step 2.

5. REVIEW - Objectives, background information, learning activities.

6. TAKE - LAP evaluation, pages 20 - 22.

7. STOP - Give your instructor the completed LAP evaluation. If you scored 85 percent or more go on to LAP 3. If not, go to step 5.

UNDERSTANDING THE LATHE

UNIT II

UNIT OBJECTIVE

After completion of this unit, the student should have a general understanding of the lathe. This knowledge will be evidenced by correctly performing the procedures outlined in the learning activities and by scoring at least 85 percent on the unit test.

SPECIFIC OBJECTIVES

After completing this unit, the student should be able to:

1. List ten safety rules for proper use of power tools.
2. Explain why a three prong grounding plug should be used on a machine such as the lathe.
3. Explain why the lathe motor may heat up and get too hot in some cutting operations.
4. List five points requiring lubrication on the lathe.
5. Explain how to level the cutting tool on the lathe and also why it must be leveled.
6. Explain what is meant by backlash in the lathe and why it is significant.
7. List four methods for holding workpieces on the lathe, and explain why each would be used.
8. Explain what feed and cut mean with reference to machining.
9. List and describe four standard lathe tools, explaining why each would be used.
10. Explain why proper relief and clearances must be gound into a cutting tool.
11. Identify the major components of the lathe.

BEST COPY AVAILABLE

Background Information

SAFETY RULES FOR POWER TOOLS

1. KNOW YOUR POWER TOOL – Read the owner's manual carefully. Learn its application and limitations as well as the specific potential hazards peculiar to this tool.
2. GROUND ALL TOOLS – If tool is equipped with three-prong plug, it should be plugged into a three-hole receptacle. If adapter is used to accommodate two-prong receptacle, the adapter wire must be attached to a **known ground.** Never remove third prong.
3. KEEP GUARDS IN PLACE – and in working order.
4. REMOVE ADJUSTING KEYS AND WRENCHES – Form habit of checking to see that keys and adjusting wrenches are removed from tool before turning on machine.
5. KEEP WORK AREA CLEAN – Cluttered areas and benches invite accidents.
6. AVOID DANGEROUS ENVIRONMENT – Don't use power tools in damp or wet locations. Keep work area well illuminated.
7. KEEP CHILDREN AWAY – All visitors should be kept a safe distance from work area.
8. MAKE WORKSHOP KID PROOF – with padlocks, master switches, or by removing starter keys.
9. DON'T FORCE TOOL – It will do the job better and be safer at the rate for which it was designed.
10. USE RIGHT TOOL – Don't force tool or attachment to do a job it was not designed for.
11. WEAR PROPER APPAREL – Avoid loose clothing or jewelry which could get caught in moving parts. Wear protective head gear to keep long hair styles away from moving parts.
12. USE SAFETY GLASSES – Also use face or dust mask if cutting operation is dusty.
13. SECURE WORK – Use clamps or a vise to hold work when practical. It's safer than using your hand, and frees both hands to operate tool.
14. DON'T OVERREACH – Keep your proper footing and balance at all times.
15. MAINTAIN TOOLS IN TOP CONDITION – Keep tools sharp and clean for best and safest performance. Follow instructions for lubrication and changing accessories.
16. DISCONNECT TOOLS before servicing and when changing accessories such as blades, bits, cutters.
17. AVOID ACCIDENTAL STARTING. Make sure switch is "OFF" before plugging in power cord.
18. USE RECOMMENDED ACCESSORIES. Consult the owner's manual. Use of improper accessories may be hazardous.
19. TURN SPINDLE BY HAND BEFORE SWITCHING ON MOTOR. This ensures that the workpiece or chuck jaws will not hit the lathe bed, saddle or crossslide, and also ensures that they clear the cutting tool.
20. CHECK THAT ALL HOLDING, LOCKING AND DRIVING DEVICES ARE TIGHTENED. At the same time, be careful not to overtighten these adjustments. They should be just tight enough to do the job. Overtightening may damage threads or warp parts thereby reducing accuracy and effectiveness.
21. It is not recommended that the lathe be used for grinding. The fine dust which results from the grinding operation is extremely hard on bearings and other moving parts of your tool. For the same reason, if the lathe is near an operating grinder it should be kept covered when not in use.

WEAR YOUR SAFETY GLASSES. foresight is better than NO SIGHT!

The operation of any power tool can result in foreign objects being thrown into the eyes, which can result in severe eye damage. Always wear safety glasses or eye shields before commencing power tool operation. We recommend **Wide Vision Safety Mask** for use over spectacles, or standard safety glasses.

CUSTOMER'S RESPONSIBILITIES

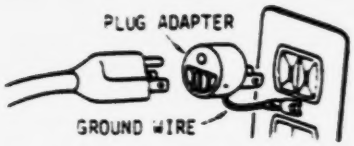
Always use care when operating the lathe. Follow safety rules for power tools. Turn off motor before attempting adjustments or maintenance. Be sure work piece is firmly supported on the lathe. Accessories should be mounted and operated following instructions carefully. DO NOT OVERLOAD MOTOR. Keep the lathe clean, lubricated and adjusted as instructed. Do not leave cleaning rags, tools or other materials on the lathe bed or around moving parts of the lathe. Install and operate lathe as directed in these instructions.

Many fine books have been written on machinery technique and are available at your local library or bookstore. Although these books will be referring to machines many times larger than a 3" lathe, the principal remains the same.

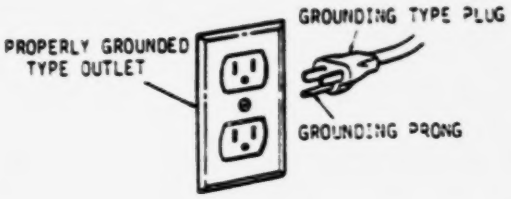
SET-UP INSTRUCTIONS

Read the instructions for cleaning, installing, lubricating and operating before attempting to install or operate the lathe.

The power cord used is equipped with a 3-prong grounding plug which should only be connected to a properly grounded receptacle for your safety. Should an electrical failure occur in the motor, the grounded plug and receptacle will protect user from electrical shock. If a properly grounded receptacle is not available use a grounding adapter to adapt the three-prong plug to a properly grounded receptacle by attaching grounding lead from adapter to receptacle cover screw.



use properly grounded receptacle as shown.



IMPORTANT

It is important to realize that you can overload the motor supplied with this lathe. The many variables involved in machining, such as materials being machined, size of cutter, shape of cutter, sharpness of cutter, diameter of stock, etc., can leave but one rule to follow . . . . . .

COMMON SENSE !

The design of the electronic circuitry which permits high torque, results in pulsating at very low RPM, but this is not harmful to the equipment.

Heavy cuts at low RPM will make the motor run hotter than lighter cuts at higher RPM. Less cooling air is supplied by the motor fan at lower RPM. In normal operation this motor will be warm to the touch, but not hot.

This does not mean that your machine cannot be run at low RPM or turn large diameters without damaging the motor, but only that the load put on the motor must be considered in each set-up and operation of the machine.

This machine is among the finest tools that may be obtained for the price, and although not offering the degree of precision found in machines costing thousands of dollars, is capable of very precise and exacting work when properly employed. In short, we feel that this is the best possible product in its price range on the market today.

90-DAY WARRANTY ON 3" METAL LATHE

If within 90-Days from the date of purchase, this Sherline 3" **Metal Lathe** fails due to a defect in material or workmanship, Sherline will repair it, free of charge.

Warranty service is available by simply contacting Sherline Products in San Marcos, California.

This warranty gives you specific legal rights, and you may also have other rights which vary from state to state.

SHERLINE PRODUCTS

BEST COPY AVAILABLE

INSTRUCTIONS

SETTING UP YOUR LATHE

With the exception of the motor and speed control and the motor bracket. the lathe is completely assembled.

Remove the rust preventative from all surfaces by wiping with a cloth moistened with mineral spirits.

For stationary use. screw the lathe to a workbench using the holes in its base. For portable use. fasten the lathe to a suitable board.

To fit the motor bracket. remove the two screws from the rear of the headstock. insert them through the holes in the bracket and screw them back into the rear of the headstock. Tighten them with the hexagon key provided.

To fit the motor to the bracket, first fit belt over motor pulley and over headstock pulley. Align the speed control bracket with the holes in the mounting lug and fit screws and washers (included in parts bag) through slots in the bracket, through speed control bracket and into motor mounting lugs. Tighten screws by hand just enough so that motor can still be moved in slots. Pull motor down slots until belt is relatively tight and tighten screws with proper hex wrench.

(Figure 2. Motor Mounting procedure.)

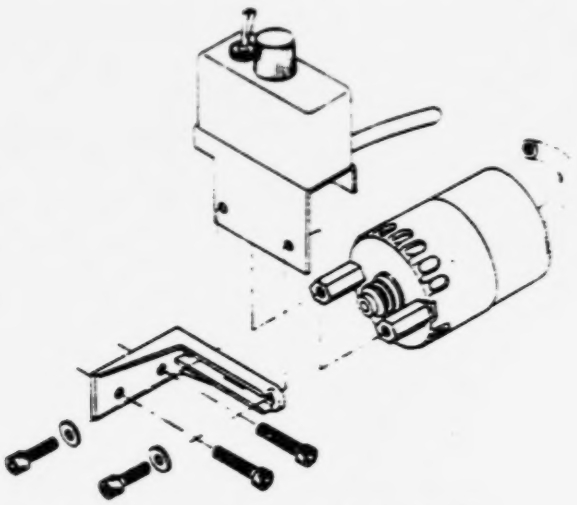


FIGURE 2. MOTOR MOUNTING PROCEDURE

LUBRICATION

BED. SADDLE. CROSSLIDE – Use a light oil such as sewing machine oil on all points where there is a sliding contract. This should be done immediately after each clean-up.

LEAD SCREW. TAILSTOCK SCREW. CROSSLIDE SCREW – Sewing machine oil should be placed along all threads regularly. At the same time check that the threads are free of any metal chips.

TAILSTOCK SPINDLE – Wind out the spindle as far as it will go and lightly oil it with sewing machine oil.

HANDWHEELS – A few drops of light oil or grease behind the handwheel will reduce friction between the surfaces and make operation easier and smoother.

HEADSTOCK BEARINGS – These bearings are lubricated at the factory for the lifetime of the machine and should not need further lubrication. Do NOT break seals.

MOTOR – Every 50 hours of running place 5 drops of sewing machine oil onto the bearing at each end of the motor.

OPERATING INSTRUCTIONS

The lathe is supplied with an electronic speed control which produces a comprehensive range of speeds suitable for all operations. Most metal turning should be done using a slow speed. For work on other materials, or for turning small diameter metal, a higher speed will be used. The electronic speed control feature allows for an increase in speed to compensate for decrease in diameter during work. It may also be used to increase the power if the cutting load affects the motor speed. The speed range of the motor using the control is from 100 RPM to 2000 RPM. This feature offers infinitely variable speed within this range without the time consuming operation of changing belts or gears as is necessary on most other lathes.

BEST COPY AVAILABLE

BEST COPY AVAILABLE

To operate the motor. turn the speed control knob counter-clockwise as far as it will go. Then turn switch to "ON" and select the speed by turning the speed control knob clockwise.

Each type of turning work requires the correct tool for the job. It is important that the cutting tool is sharp and correctly set up in the toolpost. The cutting edge of the tool should be exactly level with the center height of the lathe. Check this against either the headstock center or tailstock center. *(Figure 3. Leveling the Tool.)*

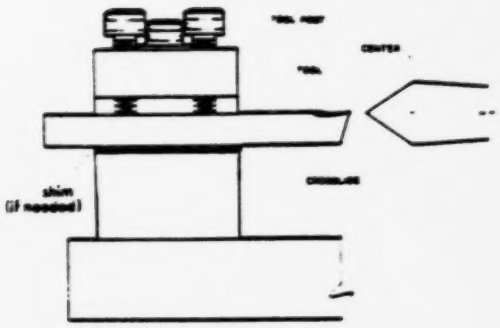


FIGURE 3. LEVELING THE TOOL

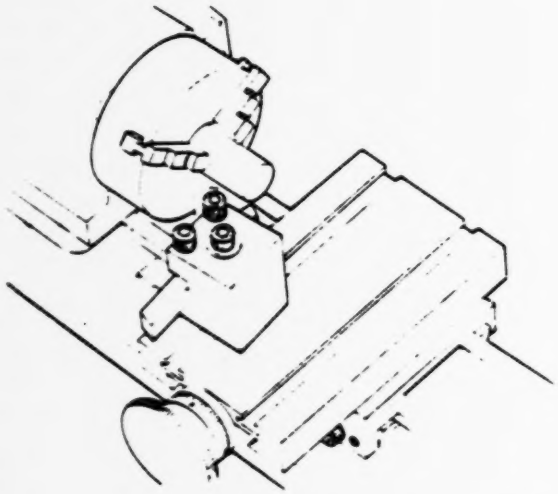


FIGURE 4. HOLDING ROUND WORK PIECE

Loosen the T-bolt and slide the toolpost as close to the center as possible. The tip of the tool bit may be raised or lowered by sliding a shim underneath it. The cutting edge must be on center or just below center (0.004" or 0.1" mm maximum). Ensure that the tool is fixed securely in position by firmly tightening the socket head screws. Try not to have the tool cutting edge protruding more than 3/8" (10mm) from the toolpost. When the tool is correctly set and the speed range selected. turn the lathe motor switch on and adjust the electronic speed control knob to give the best speed for the job. Turn the crosslide and saddle handwheels to bring the tool into position.

Turning the appropriate handwheel moves the saddle, crosslide and tailstock spindle. For American model, one complete turn of the handwheel gives a movement of 0.050". For metric model. one complete turn of the handwheel gives a movement of 1 mm. Handwheels are calibrated in 0.001" for American model and 0.01 mm for metric model. Keep the screws clean and oiled.

Back-lash. the amount of play in the lead screw, must be allowed for by feeding in one direction only. Example: You are turning a bar to .600 diameter. The bar now measures .622 which requires an .011 cut to bring it to a finished diameter of .600. If the user inadvertently turns the handwheel .012 instead of .011. he couldn't reverse the handwheel just .001 of an inch to correct the error. The handwheel would have to be reversed for an amount greater than the back-lash in the feed screws before re-setting the handwheel to its proper position.

HOLDING THE WORK PIECE

(Figure 4. Holding Round Work Piece and Figure 5. Holding Square Work Piece.)

Work can be held between centers. in three or four jaw chuck. on the faceplate. or with a collet. Sometimes it is necessary to use a chuck and center. and. if the work is spinning fast. a live center should be used.

BEST COPY AVAILABLE

TURNING BETWEEN CENTERS

This is done by fitting the dog to the work which is to be turned and placing the work and dog between the centers in the headstock and the tailstock. The maximum diameter which can be held with the dog is 5/8". *(Figure 6, Turning Between Centers.)*

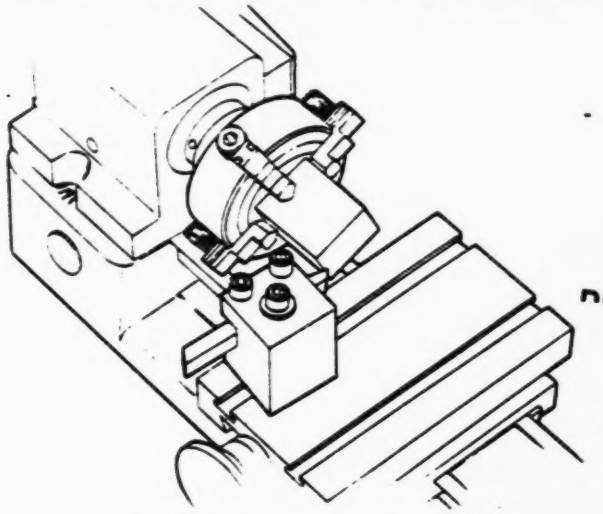


FIGURE 5, HOLDING SQUARE WORK PIECE

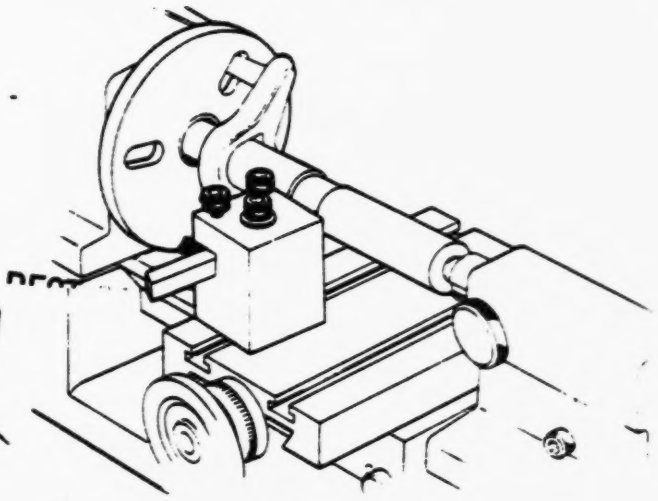


FIGURE 6, TURNING BETWEEN CENTERS

The dog is driven by fitting it into one of the faceplate holes. This method of turning is ideal for bar work or turning of steps on a bar. The tailstock center must be greased to prevent over-heating. The headstock spindle has a Morse No. 1 taper in the spindle nose. The spindle thread is ⅝-16 TPI. Accessories held in the spindle using the Morse No. 1 taper can be removed with the use of a knock-out rod (not supplied) approximately 3/8" in diameter and 6" long. The bar is inserted through the back of the spindle and accessories, such as, centers, can be removed with a few taps. The tailstock spindle is equipped with a Morse No. 0 taper and accessories such as drill chucks and centers can be removed by turning the handwheel counter clockwise until the accessory is ejected.

CENTER DRILLING

Due to the fact that the work turns and the drill does not on a lathe, it is necessary to use a center drill before a standard drill can be used. Due to the flexibility of a standard drill bit, it will tend to wander on the surface of the rotating work whereas a center drill is designed to seek the center and begin drilling. Cutting oil is recommended for all drilling operations.

TAILSTOCK DRILLING – Hold the work in a three jaw or four jaw chuck. If the work is longer than approximately 3", support the free end with a steady rest. Fit the drill chuck to the tailstock with a No. 0 Morse arbor and secure a center drill in the chuck. Adjust the tailstock to bring the center drill close to the work and lock in position. Turn the tailstock handwheel to bring the center drill forward. After the hole is started with the center drill, use a standard drill bit of the desired size to drill the hole.

HEADSTOCK DRILLING – Fit the drill chuck into the headstock using the No. 1 Morse arbor. Punch a center mark on the work with a center punch. Hold the work in the toolpost or on the crosslide. Wind the crosslide and saddle into position so that the center drill is in line with the center punch hole. Wind the saddle towards the drill chuck to drill the hole. *(Figure 7, Center Drilling, Work Turns, and Figure 8, Center Drilling, Drill Turns - See Next Page)*

INITIAL TEST CUTTING

If you have never operated a lathe before, it is suggested that you make a trial cut on a scrap material to learn the operation of the machine. Secure in a three or four jaw chuck (accessory) a piece of aluminum stock approximately ¾" diameter and 1½" long. Secure the tool supplied with the lathe in the toolpost making sure that it is properly positioned (see diagram 3). Turn the speed control all the way counter-clockwise and then turn the motor on. Bring the speed up to approximately 1000 RPM (half speed). To establish tool position in relation to the work, bring the tool in slowly until it just starts to scribe a line on the work. Crank the tool towards the tailstock until it clears the end of the work. Advance the tool .010" using the crosslide handwheel (10 divisions on the handwheel scale). Using the bed handwheel move the tool slowly across the work toward the headstock.

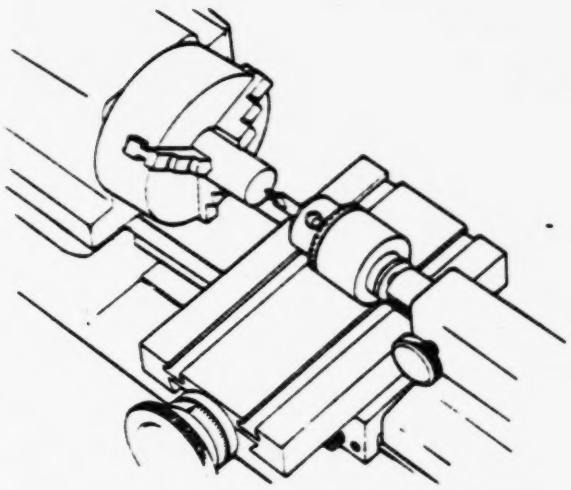


FIGURE 7. CENTER DRILLING, WORK TURNS

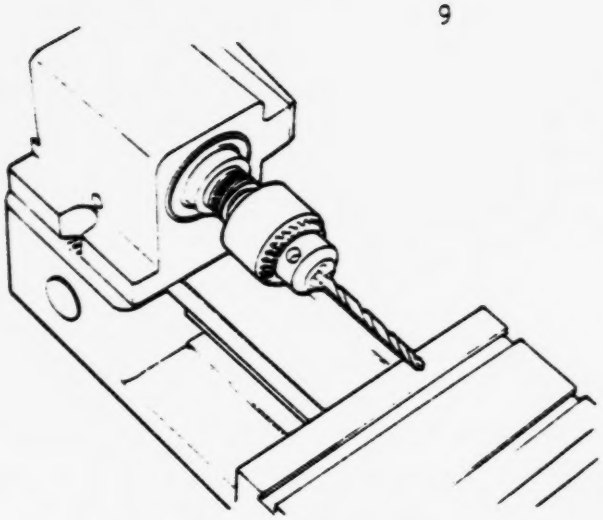


FIGURE 8. DRILLING, DRILL TURNS

If the tool is fed into the work too quickly, the main spindle will slow down considerably. Too much pressure may cause the motor to stall, but this will not do any damage if the tool is backed off immediately. If the tool jams in the work, turn off the motor and back off the tool. As long as a soft metal is used, there is little danger of damaging the cutting edge of the tool. If the swarf (removed metal) is smoking and the tool is getting very hot, then the speed is too fast or cooling compound, such as cutting oil or soluble oil, should be used.

With a little practice on scrap brass or aluminum, you will soon be able to judge correct motor speed and feed rates. If the tool is cutting material efficiently, then you are operating the machine properly.

After work has been completed, remove all shavings from feed screws and slides, clean all over, and re-oil. A well looked-after lathe will maintain precision.

ADJUSTMENTS

Gibs are fitted at the rear of the saddle and on the left hand side of the crosslide. Correct adjustment of the gibs will ensure smooth and steady operation of the slides. The gib is effectively a wedge which is adjusted by means of loosening the gib locking screws and pushing the gib in until play is removed. After adjusting, retighten the locking screws. Milling operations require a tighter adjustment of the gibs than lathe operations.

SPINDLE ADJUSTMENT – If any end play develops in the main spindle it can be easily eliminated by readjusting the preload nut (see part no. 4016 in the exploded view). Remove main pulley, loosen set screw and tighten the preload nut while turning the spindle until play is removed. Be sure to tighten it only enough to remove excessive play and no tighter, or damage may result to the bearings (0.0002" play is recommended).

REAMING

Twist drills will generally not drill perfectly accurate sizes and very small boring tools are not satisfactory in deep holes owing to their flexibility. Therefore, reaming is used for holes requiring accuracy within 0.0005" (0.013 mm). Reamers are available in any standard size but are rather expensive and are generally not purchased to do one-of-a-kind type work. Use them only when a boring tool cannot be used because of the depth of the hole.

Reamers are only used to "clean up" the hole. To make an accurate hole, the work is drilled approximately .010" smaller than the reamer size. The work should be slowly rotated and the reamer slowly fed into the hole while applying plenty of cutting oil. The reamer should be frequently removed and cleared of chips. Never rotate a reamer backwards in the work as this can dull the cutting edges.

FACEPLATE TURNING

The faceplate has three slots which allow work to be bolted to its surface. Flat work can be screwed directly to the faceplate. Extra holes can be drilled to suit odd shaped work unsuitable for a chuck. If the work is mounted off center, be sure to counter balance the faceplate, and use very low RPM.

BEST COPY AVAILABLE

TAPER TURNING

On most lathes a taper can be cut by offsetting the tailstock. On this lathe it is done by turning the headstock to any angle away from dead center. To adjust the headstock angle, loosen the screw on the front of the headstock and slide out the square key just protruding on the left hand side below the headstock. The headstock can now be set for the required taper by lining the headstock mark to the required angle and tightening the screw. The base is calibrated in 5° increments up to 45° both left and right of dead center. *(Figure 9, Taper Turning).*

Short work can be inserted in a three or four jaw chuck and turned as shown in the drawing. To machine a taper in longer stock, center both ends of the bar, set the headstock angle and mount between centers. If the headstock is angled towards the lathe front, the taper will cut smaller at the right. Tapers can also be bored in work held in the three or four jaw chuck.

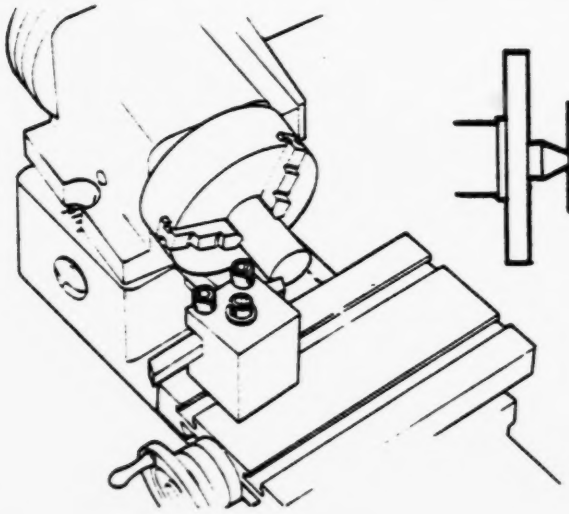


FIGURE 9, TAPER TURNING

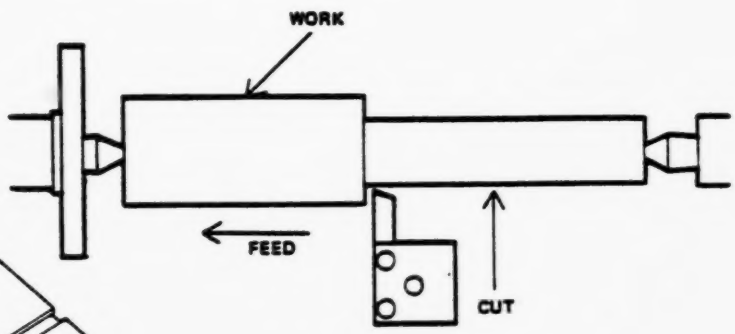


FIGURE 10, FEED AND CUT

GENERAL MACHINING

Two terms frequently used are "Feed" and "Cut". Reference to the diagrams will show what is meant by these terms. Normal turning, when used to reduce the diameter of a work piece, involves advancing the cutting tool **across** the lathe bed by an appropriate amount (depth of cut) and then **feeding** the tool along parallel to the lathe bed to remove material the desired length. *(Figure 10, Feed and Cut).*

In this case the depth of cut is set by the crosslide handwheel, and the feed is provided by the handwheel on the end of the bed. When facing off a work piece held in a chuck or faceplate, the depth of cut is set by the handwheel on the end of the bed, and the feed is provided by the crosslide handwheel. *(Figure 11, Facing Off - See Next Page)*

Cutting tools are ground to various shapes according to their usage. Tools are usually ground to shape as needed by the operator. Some standard tools are described below:

(a) **Normal Turning Tool** – This feeds from right to left, is used to reduce work to the desired diameter and is the most frequently used of all tools.

(b) **Side Tools** – These are used to face off the ends of shoulders and may also be used as normal turning tools. Note that one of these is then fed from LEFT to RIGHT and is called a LEFT side tool.

(c) **Parting Tool** – This is shaped like a dove-tail when viewed from above and is used to cut off work pieces by feeding the end of the tool across the lathe bed and through the work piece. Note that feed and cut become synonymous with this tool.

(d) **Boring Tool** – Used to enlarge holes in a work piece.

The shape and use of these tools is illustrated.

(Figure 12, Boring Tool, Figure 13, Tools). see next page

The clearances ground behind the cutting edges indicate the type of material for which the tool may be used and the direction in which it is fed along the work. When grinding tool bits, correct clearances are essential or "rubbing" can occur.

BEST COPY AVAILABLE

BEST COPY AVAILABLE

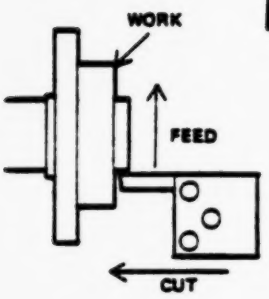


FIGURE 11. FACING OFF

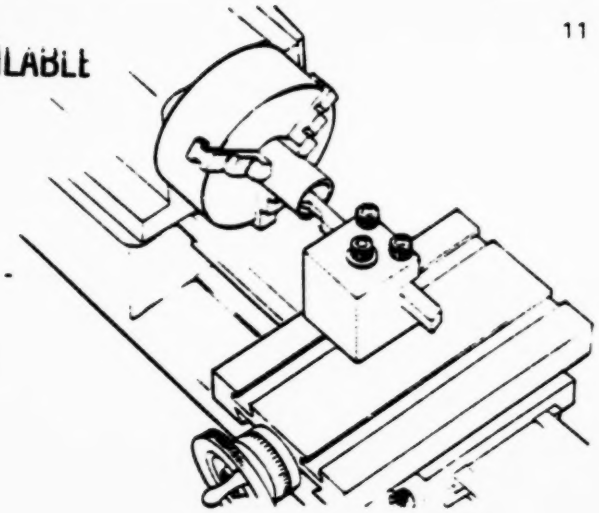


FIGURE 12. BORING TOOL

Turning Tools (left and right) – Reference to Figure 13 will illustrate the lateral positioning of this tool. Note the clearance behind the point between the end of the tool and the work. Insufficient clearance will cause the tool to "rub" and excessive clearance will produce a ridged or wavey finish due to the small length of tool edge in contact with the work. This ridging becomes more pronounced with rapid feed. To provide a smooth finish the sharp cutting point may be slightly rounded with an oilstone, taking care to preserve the side clearance underneath this corner. *(Figure 14).*

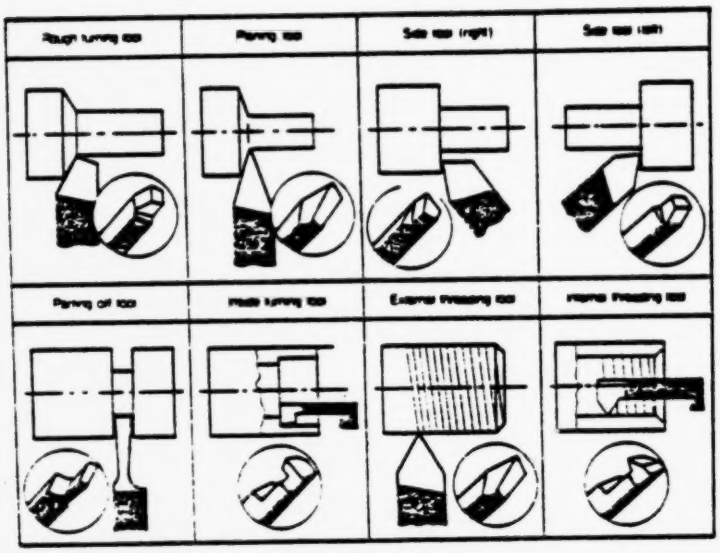


FIGURE 13. TOOLS

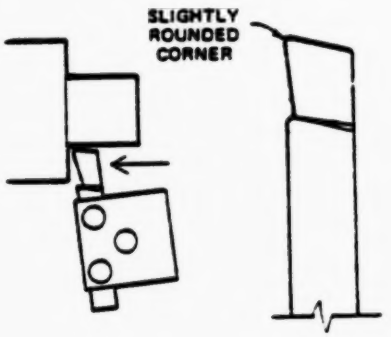
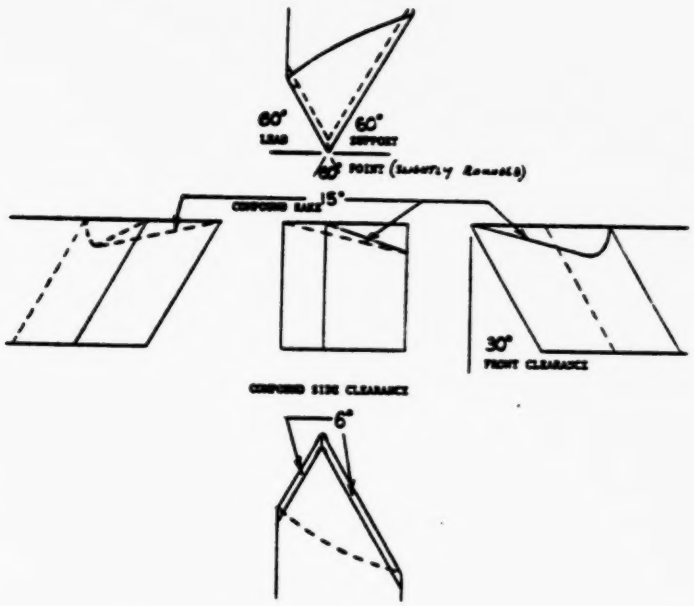


FIGURE 14. ARROWS SHOW DIRECTION OF TOOL FEED IN ALL DIAGRAMS

This tool should not be advanced directly endways into the work, the depth of cut being set while the tool is clear of the end of the work. The starting procedure is to advance the tool until the point just touches the work, note the reading on the crosslide handwheel, withdraw tool slightly and move along until clear of the end of the work. Now advance the crosslide to the above reading, add desired depth of cut, and then feed the tool along the work piece the desired distance. Withdraw the tool clear of the work, having noted the reading on the crosslide handwheel, mentally note the reading on the lead screw handwheel, return tool to starting position, and advance to the previous reading plus the desired cut.



Lathe Tool Grinding Procedures

1. Grind 30° front clearance. This provides the actual cutting point. If too much angle is placed here insufficient support is provided. If too little angle is provided the tool tends to be blunted against its workpiece. In either case premature failure will result. To achieve the 30° front clearance the tool rest should be set at 15° relative to a horizontal plane (check this with a protractor).
2. Grind 60° lead and support angles. This should be gauged at the top with a center gauge, and laid-out for lines to work toward. As shown on the diagram, the lead and support sides are not equal in length/size, and the point is off-center. The lead side is shorter and stronger, and thus, is used as the leading edge in the cutting action. Conversely, the support side should be longer to accept the compound rake angles. Both the lead and support are ground using the same 15° tool rest angle as was used on the front clearance. When ground using the 15° tool rest angle, following the laid-out lines, the 6° compound side clearance will be accomplished.
3. When completed, the 60° point should be 1/3 on the lead side and 2/3 on the support side.
4. Using the right side of the wheel, grind the top rake at a 15° compound angle. The 15° compound angle is necessary for curling and efficiently removing the chips. Be sure to slant the compound rake back and over simultaneously, all in one cut, on the grinding wheel. A smooth rake will assist in efficient chip removal, while an irregularly ground rake will provide resistance to chip flow, leading to built up edge (BUE).
5. Using the honing stone make the point, from top to bottom, rounded (a slight radius). This will assist in dissipating heat, rather than localizing the heat. Again built up edge will less likely occur and tool life will be increased.

The Basic Cutting Tool.

BEST COPY AVAILABLE

The second feed is now commenced, ceasing at the same previous reading of the lead screw handwheel, after the appropriate number of complete turns of this handwheel. This procedure enables turning to accurate length.

Repeat the procedure until the work has been reduced to within about 0.010" (0.25 mm) of desired diameter, noting that each 0.015" (0.4 mm) increase in depth of cut will reduce the work diameter by twice this amount, i.e. 0.030" (0.8 mm). For the finishing pass, advance the tool by the required amount and feed along the work just far enough to gauge the finished diameter. Adjust depth of cut if necessary and complete the final pass using a SLOW feed to obtain a smooth finish and exact size. Filing in the lathe can be a dangerous operation if not carried out correctly.

Parting Tool – This tool will cause no end of trouble if it is not correctly ground, set and used. Setting to exact center height is essential and exact lateral setting requires that the sides behind the broad cutting edge will clear the sides of the groove in the work as the tool advances. (*Figure 13, Tools*).

The tool should be clamped in the tool post with a minimum of overhang to avoid "spring" and consequent tool chatter and digging in. The cutting off operation should be carried out with the tool positioned as close to the chuck or center as possible, to avoid "spring" of the work. Maximum rigidity and slow speed are essential. A small square held against the work will assist in locating the tool squarely. If the tool chatters or produces noise as it is fed into the work, try reducing the turning speed, as a parting tool has much greater length of cutting edge in contact with the work than most other tools. An alteration in the rate of feed will also assist.

Feed the tool positively and evenly into the work so that it neither digs in nor begins to rub. Watch that the sides of the tool clear the sides of the work as it advances. NEVER attempt to completely cut off a work piece when held between centers as this is positively dangerous. Stop while there is still a reasonable connecting piece, remove the material from the lathe and break off or saw off. The work may be chucked to turn off any projection left on the end.

Very small work may be completely cut off when held in a chuck, and allowed to fall onto the crosslide as it is too small and light to cause any damage. Hollow articles, e.g. rings, may be caught on a piece of wire whose end is held in a suitable position.

Side Tools – While these may be, and often are, used as general purpose turning tools, their specific use is for facing the sides of collars and shoulders (i.e., finishing these to correct dimensions and smooth flat surface), and for facing work held on a faceplate or in a chuck. The facing of work in this manner is very useful for the production of truly flat surfaces and for producing articles to an exact thickness. The two uses of the side tools are illustrated in Figure 16, Side Tools. The sharp corner at the cutting point should not be slightly rounded, as may be done with the normal turning tool, as knife tools may be required to produce sharp corners.

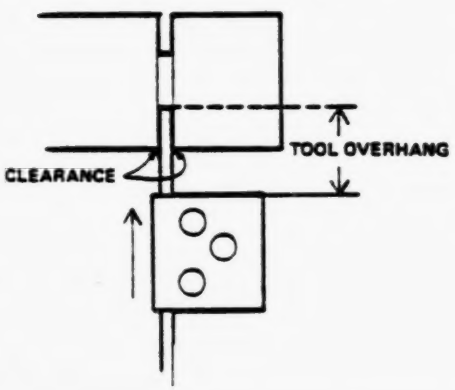
BORING TOOL

The use of this tool requires the existence of a drilled or cored hole, or it may be used to enlarge the bore of a tube. The work must be mounted in a chuck or on a faceplate and the boring tool set as shown in Figure 17, noting the clearance behind the cutting point.

A slow rate of feed should be used, as the turnings are not able to escape freely from the hole. Frequent withdrawal of the tool to allow turnings to escape may be necessary. Care should be taken not to feed the tool beyond the depth required, or to feed so deeply as to damage the chuck or faceplate.

Where a hole must be bored right through the work, it should be shimmed out from the faceplate to provide clearance for the tool to feed through. The lead screw handwheel graduations may be used to indicate the correct depth at which to stop the feed. Notice that with boring, the depth

FIGURE 15, PARTING TOOL



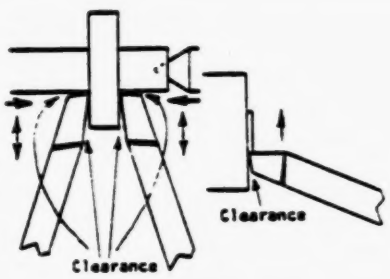


FIGURE 16, SIDE TOOLS

FIGURE 17, BORING TOOL CLEARANCE

BEST COPY AVAILABLE

of the cut is increased by moving the tool and crosslide towards the operator and not away as with normal turning.

The boring of holes often necessitates greater than normal overhang of the tool from the tool post, so the depth of cut and rate of feed should be reduced from normal.

TOOL CUTTING ANGLES

The shaping of cutting tools to suitable angles for the type of material and nature of work being performed can be very important to satisfactory work. When tools become dull, gently re-grind and preserve the original angles and shapes. Better cutting and longer life will be obtained from cutting edges if they are finished with a small oilstone to obtain a really keen, smooth edge. **Do Not Grind The Top Face** of the tools, but confine sharpening to the end and/or sides.

Remember that heavy cut and rapid feed will cause greater strain on the chuck and lathe, may induce "spring" or binding of work and tools, and also produce a poor finish.

TURNING SPEEDS

The chart below provides a guide to approximate speeds at which work of differing materials should be rotated. Note that the turning speed is inversely proportional to the diameter of the work: the larger the diameter, the slower the turning speed. Material often differs in its hardness, so these figures may have to be varied. The harder the material, the slower the turning speed should be.

GUIDE TO APPROXIMATE TURNING SPEEDS

MATERIAL	¼" (6 mm) DIAMETER	½" (13 mm) DIAMETER	1" (25 mm) DIAMETER
Annealed Carbon Steel	800 RPM	400 RPM	200 RPM
Cast Iron	900 RPM	450 RPM	225 RPM
Mild Steel	1500 RPM	750 RPM	375 RPM
Brass	2300 RPM	1150 RPM	575 RPM
Aluminum Alloy	3000 RPM	1500 RPM	750 RPM

Keep in mind that, apart from possible production of excessive heat (which will cause the work to expand) and the fact that excessive speed may damage the cutting edge or cause it to "rub" instead of cutting, turning speeds are not too critical. Slower speeds than normal cause no harm, except by increasing the time involved. Aluminum, however, usually gives a better finish turned at high speed and lubricated.

LUBRICANTS

Much can be written about the use of lubricants, but they may usually be dispensed with where production rates are not very important. Aluminum and its alloys may require the use of turpentine, paraffin oil or kerosene to prevent the chips from welding to the tool's point. If desired, a mixture of one part soluble oil to six parts water may be used on steel to assist in producing a smoother finish and reduce tool chatter when parting off. Brass and cast iron are always turned dry. Lubricants should be removed from the lathe after use.

The main purpose of using lubricants is to remove heat from the tool and the work. Modern high speed tool bits are not likely to be affected by heat on the type of work usually done on this lathe.

USE OF ACCESSORIES AND ATTACHMENTS

Your lathe can be made more versatile with the addition of suitable attachments and accessories. These include various chucks, a screw-cutting attachment, a milling column, collets, a live center and many others. Some of the most popular ones and their uses are listed here.

Remember that accessories and attachments must be cared for in the same way as the lathe. Always make sure that threads are free from metal chips and dirt. Chucks should be lightly oiled frequently so that they continue to function smoothly and accurately. Gears in the screw-cutting attachment should be lightly greased when in operation. Some attachments have moving slides and these should be lubricated in the same way as the slides in your lathe (see "LUBRICATION").

BEST COPY AVAILABLE

THREE JAW CHUCK

The three jaw self-centering chuck is the most popular of all lathe attachments. This chuck will grip round or hexagonal work quickly since the jaws move simultaneously to automatically center the work being held. The jaws on the chuck are designed so that the same chuck can be used for both internal and external gripping. Jaws are reversible for holding larger diameter work.

Due to the nature of the design of a three jaw chuck, it cannot be expected to run perfectly true. Even three jaw chucks costing 5 times as much as the one made for this lathe will have 0.002" to 0.003" runout. If perfect accuracy is desired in a particular operation, the use of a four jaw chuck is recommended.

FOUR JAW CHUCK

Each of the four reversible jaws on the four jaw chuck can be adjusted independently of the others. It is used widely because of its capacity to hold almost any shape of material in a centered, concentric or eccentric position, and it can be adjusted to whatever degree of accuracy is required.

The most convenient way of offsetting work requires use of a dial indicator. The work is first centered roughly, and then the chuck is revolved by hand to determine in which direction the work must be moved. Adjustments may require loosening and tightening of two or more jaws at a time. The final centering needed shows on the dial indicator, which indicates movement under the tip in micrometer measurements. Another method of centering is to use a thin feeler gauge or paper as a center finder. Mount a blunt ended bar in the toolpost. Place the feeler or paper strip between the bar and the workpiece and advance the crosslide until the tension just holds the paper in place. Then rotate the chuck by hand, and if the tension increases or decreases, it indicates that the work is not centered. Another very useful and precise method is to coat the work with layout stain. Rotate the work by hand and a lathe toolbit will mark the high spot on the work.

For obvious safety reasons, off-center or out-of-balance work should be turned at the slowest possible speed.

DRILL CHUCK

The drill chuck is also one of the more popular attachments. It is used mainly for drilling from the tailstock. When the lathe is to be used for drilling, the work can usually be mounted in one of the other lathe chucks, or clamped to the faceplate for attaching to the main spindle. It is important to remember that when drilling, a center drill is used to start the hole. Then the hole will be drilled concentrically with the spindle. This is particularly important when drilling deep holes with small diameter drill bits.

A No. 1 Morse arbor is available for the drill chuck so that it can be fitted into the main spindle.

All chucks must be kept clean. The three and four jaw chucks should be washed in solvent and oiled lightly when new and after use. The drill chuck should be washed in solvent and oiled occasionally.

COLLET ATTACHMENT AND COLLETS *(Figure 18, Collet Attachment and Collet).*

A collet is essentially another type of chuck used when greater centering precision is required with smaller diameter round bars than is possible to obtain with the three jaw chuck. The collet set available for this lathe is designed to use a standard WW collet (Watchmaker's Collet). The set includes a spindle-to-collet adapter, a draw bar and five collets: 1/16", 1/8", 3/16", 1/4", and 5/16". A complete range of collets in both metric and American sizes are available through most industrial suppliers.

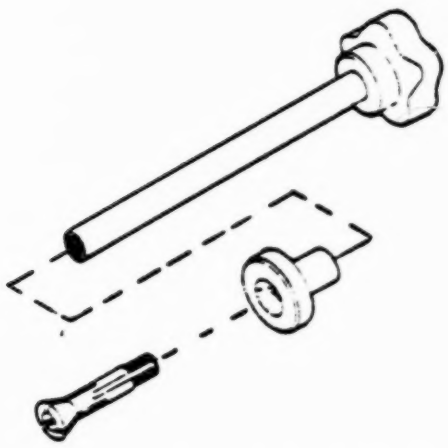


FIGURE 18. COLLET ATTACHMENT AND COLLET

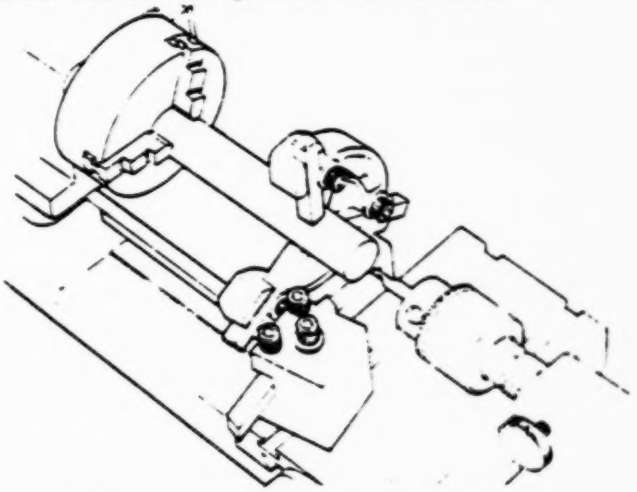


FIGURE 19. 3 POINT STEADY REST

THREE POINT STEADY REST *(Figure 19, 3 Point Steady Rest - See Preceding Page)*

A steady rest is needed to prevent whipping or wobble when turning slender or fragile bars, or drilling into the end of long cylindrical pieces. The quickest way to set up the rest is to support the work between centers or between chuck and tailstock. Loosen the three set screws with the correct hexagon key and push the brass bearings inward until they contact the work lightly. Lock in position and lubricate the bearings with oil or grease. Never run them dry. At the point of contact, the work must be smooth. Brass bearings are used because they cause minimum marring of the work.

Another advantage of the steady rest which is often overlooked is the fact that work which is held in position by the rest must turn concentrically with its outside diameter. This means that concentricity is assured when working near the steady rest because at that point it MUST be running perfectly true despite imperfections in the way it is chucked or centered at either end.

THREAD CUTTING

Three methods of cutting threads on the lathe are with taps, dies or the screw cutting attachment. Taps and dies are more economical when only two or three thread sizes are required. The screw cutting attachment, however, offers the possibility of cutting a wide variety of threads, both metric and American, with left or right hand leads. Full instructions are provided with the attachment set so that even a novice can cut threads without difficulty.

The screw cutting attachment works on the principle of coupling the main spindle to the lead screw with a gear train. Threads are cut by having the saddle moved at a fixed rate in relation to the main spindle. Internal and external threads can be cut. Various thread shapes can be cut depending upon the angle to which the tool is ground. When cutting threads the motor is removed from the lathe, and a hand crank is connected to the spindle. The work is then rotated by hand with the crank for better control. *(Figure 20, Thread Cutting Attachment).*

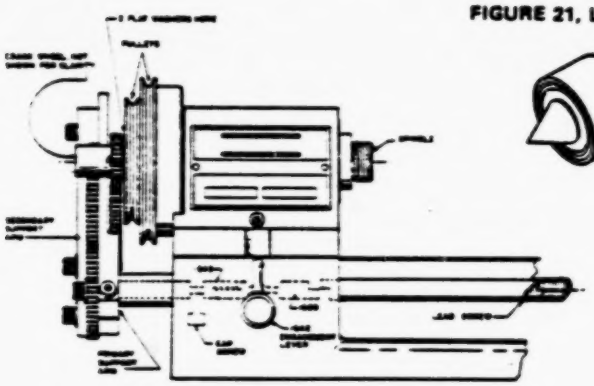


FIGURE 20, THREAD CUTTING ATTACHMENT

FIGURE 21, LIVE CENTER

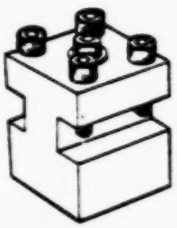


FIGURE 23 TWO POSITION TOOL POST

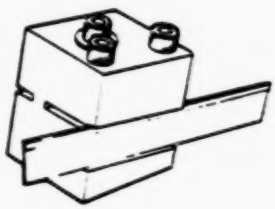


FIGURE 22, CUT-OFF TOOL AND HOLDER

To cut internal threads with a tap, first prepare the correct size hole for the desired thread. Drill manufacturers supply charts of drill sizes which will give this information. The tap is wound into the rotated work by hand, with plenty of oil lubricating the cutting edges. External threads are cut with a die. Gently turn the work by hand and use a liberal amount of oil on the die. If the die starts to bind, back off, brush the chips away and take up again.

LIVE CENTER *(Figure 21, Live Center).*

The live center rotates with ball bearings, eliminating much friction and the heat it generates – a problem especially with plastics.

CUT-OFF TOOL AND HOLDER *(Figure 22, Cut-off Tool and Holder).*

The cut-off tool and holder were designed to eliminate many of the problems in using a parting tool. The holder assures the proper alignment of the tool for accurate parting. The tool itself is only .040" wide and makes a fast, accurate cut through the work.

TWO POSITION TOOL POST *(Figure 23, Two Position Tool Post).*

Extra tool holders are inexpensive and can save much set-up time by leaving more than one tool mounted and set up, ready to use. It is much quicker to change tool holders leaving the tool properly positioned in its own holder than to change tools in the holders.

POWER FEED *(Figure 24, Power Feed).*

The power feed incorporates a separate gear motor to drive the saddle at the proper speed for achieving an excellent finish on the part, and it eliminates much tedious hand cranking. Feed is from right to left only, at approximately 1" per minute. The power feed motor is left running and is engaged with the lead screw by a lever installed directly on the headstock. Complete installation instructions are provided with the attachment.

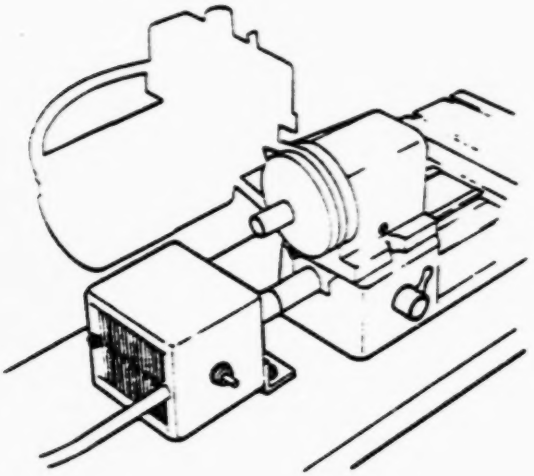


FIGURE 24. POWER FEED

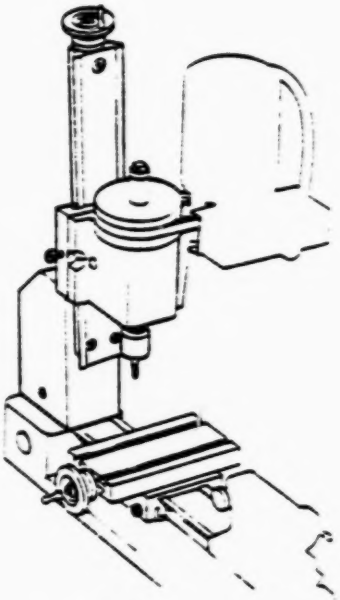


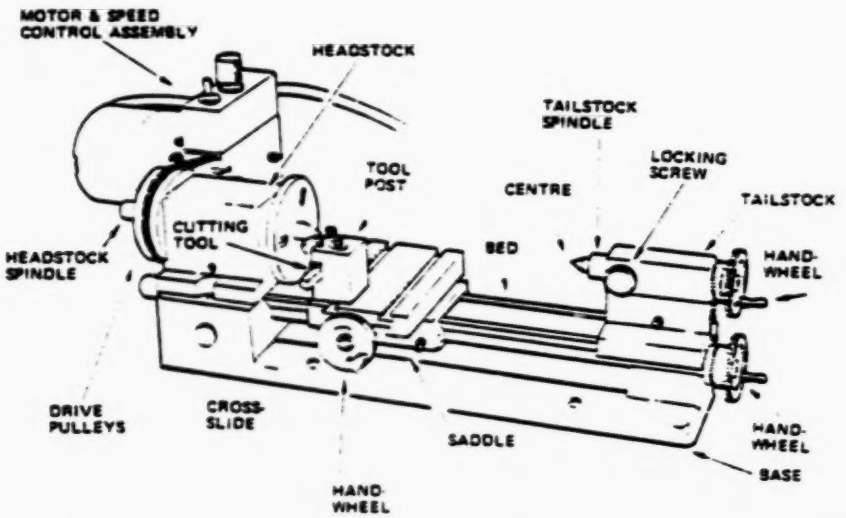
FIGURE 25. VERTICAL MILLING ATTACHMENT

VERTICAL MILLING ATTACHMENT *(Figure 25, Vertical Milling Attachment).*

The milling attachment is made with a solid aluminum base and dovetailed vertical column. The column has a saddle which can be raised or lowered by rotating the handwheel. The milling column is fitted to the left side of the lathe where the headstock is normally attached. Work is usually fitted to the crosslide and held in a chuck, or directly on the slide with clamps or in a vice.

Cutters are held in the headstock by a chuck, collet or special cutter holder.

Grooving, keyway cutting and flycutting are some of the different types of milling operations possible.



BEST COPY AVAILABLE

UNDERSTANDING THE LATHE
UNIT II
LEARNING ACTIVITIES

DIRECTIONS

The following activities are provided to allow you to reinforce your learning of Unit II. Complete each activity exactly as prescribed and do not proceed to Unit III until all activities for Unit II have been accomplished.

1. Explain in your own words, three operations which can be performed on the lathe; (a) _____

   _____

   (b) _____

   _____

   (c) _____

   _____

2. Explain, in your own words, why you think the lathe is an important machine tool in industry _____

   _____

   _____

   _____

3. Remove the cutting tool from the lathe, examine it for wear and proper cutting geometry. If it needs "cleaned up", work with your instructor to accomplish this, making sure the instructor checks it prior to attempting to use it. Put the tool in the machine and level it.
4. Set up the machine to enable a standard facing and turning operation using a piece of .500 inch brass round stock, Manually remove a total of .250 inch from the stock, one inch along its length.
5. Explain, in your own words, the general functioning of the lathe. In other words, how does it actually remove metal? _____

   _____

   _____

   _____

   _____

UNDERSTANDING THE LATHE
UNIT II
EXPECTED RESULTS OF LEARNING ACTIVITIES

1. (a) General machining - This is simply removing metal along the length of the workpiece, changing its diametral dimension in the process.
   (b) Parting - Actually seperating a single piece of materialinto two components by cutting entirely through the workpiece on the lathe.
   (c) Boring - Creating a hole larger than standard drill bits will permit.
   (d) Drilling - Using the lathe as a drilling machine to generate standard holes.
   (e) Threading - Creating standard threads by using the precise cutting logic of the lathe.

2. The lathe is an important machine tool in industry because it is capable of precisely producing a variety ofcomponents based on round shapes. These components are essential for various types of technological and industrial products/activities including motors, engines, machines and so on.

3. The cutting tool should be sharp and properly leveled for operation.

4. A brass workpiece with dimensions (cut) ofone inch long by .250 inch diameter should be produced.

5. The lathe removes metal by revolving a workpiece by a stationary cutting tool.

UNDERSTANDING THE LATHE
UNIT II
LAP EVALUATION

Name _____

A. Listed below are terms, phrases and statements frequently used in understanding the lathe. Examine each statement closely and determine whether it is true or false. Indicate your answer on the line preceeding each statement.

1. _____ Three pronged plugs are useful only for lathes, not other tools/machines.
2. _____ Safety goggles are not really necesssary with the CNC lathe we are using since it is smaller than most machines in industry.
3. _____ Larger diameter cuts require fewer RPM's.
4. _____ The headstock bearings on the CNC lathe require weekly lubrication.
5. _____ Leveling the tool on the lathe is accomplished by aligning the tool point with the cross slide handwheel.
6. _____ The most accurate workholding mechanism on the lathe is the collet system.
7. _____ Center drilling on lathe workpieces is accomplished by standard drill bits.
8. _____ Reaming is a process used to enlarge a standard drill size hole.
9. _____ On most lathes a taper can be cut by offsetting the tailstock.
10. _____ "Cut" refers to how fast metal is removed.
11. _____ Side tools can be used to face off the ends of shoulders and may also be used as normal turning tools.
12. _____ Bored holes require an existing drilled hole.
13. _____ Relief and clearance in cutting tools are used primarily to reduce wear on the tool.
14. _____ The three jaw chuck is used primarily for rapid positioning/ holding of round pieces.

B. Listed below are questions requiring short answers. Read each question and then fill-in your answers on the blanks provided.

1. List ten safety rules to observe for proper use of [illegible] tools.
   a. _____
   b. _____
   c. _____
   d. _____
   e. _____

f. _____

g. _____

h. _____

i. _____

j. _____

2. Explain why a threeprong grounding plug should be used on a machine such as a lathe.

_____

_____

_____

3. Explain why the lathe motor may heat up and get too hot in some cutting situations.

_____

_____

_____

4. List five points requiring lubrication on the lathe.

a. _____

b. _____

c. _____

d. _____

e. _____

5. Explain how to level the cutting tool on the lathe,and also why it must be leveled.

_____

_____

_____

_____

6. Explain what is meant by backlash in the lathe and why it is significant.

_____

_____

_____

_____

7. List four methods for holding workpieces on the lathe, and why each should be used.

a. _____

_____

b. _____

_____

c. _____

_____

d. _____

_____

8. Explain what feed and cut mean with reference to machining.

a. Feed _____

_____

b. Cut _____

_____

9. List and describe four standard lathe tools, explaining why each would be used.

a. _____

_____

b. _____

_____

c. _____

_____

d. _____

_____

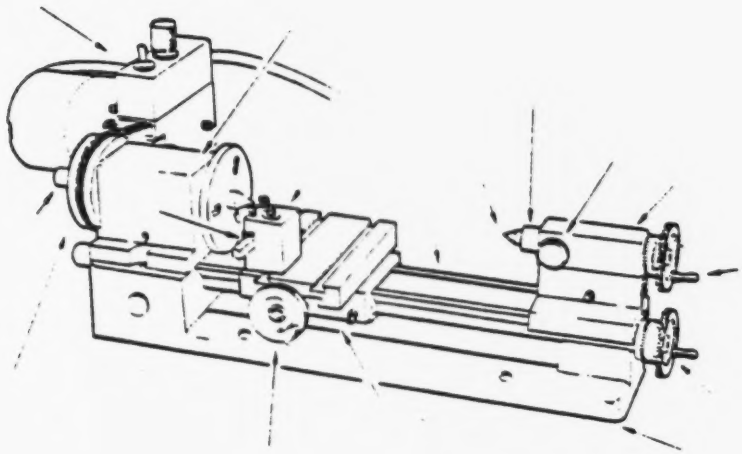
10. Explain why proper relief and clearances must be ground into a cutting tool.

_____

_____

_____

11. Identify the major components of the lathe on the drawing provided.



UNDERSTANDING THE LATHE
UNIT II
LAP EVALUATION KEY

A.
1. False
2. False
3. True
4. False
5. False
6. True
7. False
8. True
9. True
10. True
11. True
12. True
13. False
14. True

B.
1. (see page for the complete safety listing).
2. In the event an electrical short develops, the middle prong will provide an appropriate ground circuit, protecting the user from electrical shock.
3. If the motor is run too slow (RPM) for the depth of cut (too deep), the amount of work being completed becomes too great. Further, the fan in the motor must rotate sufficiently to pull enough cool air through the motor.
4. A. Bed, saddle, crosslide
   b. lead screw, tail stock screw
   c. tailstock spindle
   d. handwheels
   e. motor
5. The cutting tool point should be aligned just below the tailstock center. It must be properly aligned in order to cut properly.
6. Backlash is the normal "slop" or wear which occurs in mechanical machinery. For example, the lathe lead screws wear and cause some inaccurancy in the handwheel readings at the dial.
7. a. Centers - This is useful for gaining concentricity off the two center holes.
   b. Three Jaw Chuck - Permits quick and automatic alignment on rounds.
   c. Four Jaw Chuck - Will accept other than rounds and also can be centered.
   d. Collet - quick change mechanism providing optimum precision.
8. a. Feed - The cutter moving past or along the workpiece at a certain rate.
   b. Cut - Expresses the depth of cut into the workpiece.
9. a. Knurling Tool - For a grip or enlargement of a cylindrical surface.
   b. Parting Tool - To seperate a piece into two pieces.
   c. Standard Cutting Tool - For general cutting/turning work.
   d. Boring - Creating a larger hole from a drilled hole.
   e. Drill Chuck - Standard Drill attachment.
10. Proper relief and clearance must be ground into the cutting tool so that maximum cutting efficiency and tool life can be obtained.
11. (Diagram of lathe with labels [illegible]

UNIVERSITY OF
NORTH DAKOTA
DEPARTMENT OF INDUSTRIAL TECHNOLOGY

HIGH TECHNOLOGY MOBILE LABORATORY

Learning Activity Packet

MAJOR INSTRUCTIONAL UNIT: COMPUTER AIDED MANUFACTURING

LEARNING ACTIVITY PACKET (LAP) NUMBER: III

LAP TITLE: Using the Computer

PREREQUISITE LAP (S) : None

LAP PURPOSE: This LAP should help the student gain an understanding of the computer and how to interact with it.

COMPETENCIES TO BE DEVELOPED:

After completion of this unit, the student should be able to:

1. Identify and explain the major computer terms.
2. Satisfactorily interact with the computer to enable him/her to operate the Paxton-Patterson CNC lathe.

LEARNING STEPS
USING THE COMPUTER

1. READ - Unit objective and specific objectives, page 3.
2. STUDY - Background information, pages 4 - 20.
3. DO - Learning activities, pages 21 - 22.
4. STOP - See your instructor for evaluation. If your work is satisfactory, go to step 5. If not, go to step 2.
5. REVIEW - Objectives, background information, learning activities.
6. TAKE - LAP evaluation, pages 24 - 25.
7. STOP - Give your instructor the completed LAP evaluation. If you scored 85 percent or more go on to LAP IV. If not, go to step 5.

USING THE COMPUTER
UNIT III

UNIT OBJECTIVE

After completion of this unit, the student should be able to understand the computer and how to interact with it. This knowledge will be evidenced by correctly performing the procedures outlined in the learning activities and by scoring at least 85 percent on the unit test.

SPECIFIC OBJECTIVES

After completing the unit, the student should be able to:

1. List and explain the major computer terms.
2. Perform cassette saving and loading functions.
3. Produce a program.
4. Correct/edit the program.

USING THE COMPUTER

UNIT III

BACKGROUND INFORMATION

Using the Computer

This computer is produced by one of the leading companies in the micro-computer field . . . The Tandy Corporation.

You will find this computer offers even more capability than the standard micro-computer.

We have added two very important features that are not available on standard models. They are:

- 64K RAM memory
- Video monitor jack

The upgrade to 64K memory means you will be able to write long part programs if you choose. You will have 999 blocks of information available for part programming. The video monitor jack enables us to use a high resolution monitor – not an ordinary T V. The monitor gives a sharper image, plus the green color is less fatiguing on your eyes.

Common Terms

Here are some frequently used terms and their definitions.

RAM MEMORY - Random access memory is the computer memory where programs are stored while the computer is turned on. Turning off the computer completely erases everything in the RAM memory.

ROM MEMORY - Read only memory is the built in memory which stores programs the user does not manipulate. ROM memory in your computer contains the extended color basic language and ROM memory in the electronic motor controller cartridge contains the logic for controlling the stepper motors.

MICROPROCESSOR - The microprocessor is the central processing unit (CPU) where all the "thinking" is done. It is the microprocessor which uses all the data input to produce the desired computer output . . . a finished part or video graphics.

BIT - A Binary notation that is a basic element of a computer which electronically carries the binary 1 or 0. Eight bits compromise one byte of information.

BYTE - A series of computer binary bits organized together to hold a symbol, letter, or number in a computer.

What Language Does the Computer Use?

Talking to your computer is easy because it uses extended color basic language. You've probably heard of basic, beginners all-purpose symbolic instruction code. Extended color basic is a commonly used language, and is easy to learn.

You'll soon see this language is easy to grasp because it uses ordinary words. If you want to run a part program you just type RUN. If you want to edit something you just type EDIT and a block number. That's the beauty of basic language . . . it is easy.

Getting Ready to Program

Make sure you followed all the assembly instructions and have correctly connected all components in the CNC TRAINING CENTER. Power up the system by turning on the switch on the power strip. The following message should appear.

```
EXTENDED COLOR BASIC 1.0
COPYRIGHT (C) BY TANDY
UNDER LICENSE FROM MICROSOFT
OK
```

If this message does not appear, check to make sure the computer and monitor are turned on and properly connected.

Loading a Program

In order for the CNC TRAINING CENTER to function properly you must load the "Master Program" into the computer. The Master Program is the cassette software which allows the entire CNC TRAINING CENTER to function. The Master Program must always be loaded before the part programs. Once loaded, the Master Program stays in the memory until the computer is turned off. You may load numerous part programs throughout the day. Each time a new part program is loaded from cassette the old part program is "dumped" from the memory.

If you are inputting a new part program from the keyboard you must type NEW before beginning. This removes any other part program.

Loading a program from tape is easy. Just follow these instructions:

1. Set volume control at 5 and load cassette into recorder.
2. Press the play button button on the recorder until it locks down.
3. Type CLOADM for the Master Program (CLOAD for a part program).
4. Press enter.

The computer will search for the master program. While it is searching the letter "S" will appear in the upper left hand corner of the screen. When the computer finds the program, the letter "F" and the file name will appear at the top of the screen. The master program takes approximately 30 seconds to load. When the master program is properly loaded "OK" will be displayed on the screen directly beneath the file name.

If it doesn't load properly you may get one of these error messages:

? FM ERROR This means "Bad File Mode". You probably typed CLOAD instead of CLOADM. Try rewinding the tape and typing CLOADM.

? IO ERROR This means "Input Output Error". You will get this error if the tape was not completely rewound. Try rewinding the tape and typing CLOADM.

Note: *After loading be sure to remove the master cassette. Do not record on the master cassette tape.*

How to Program a Part

Part programming actually begins away from the computer. The programmer must evaluate the part drawing and determine step by step the path the tool must follow in order to generate the desired part contour. Creating part programs from drawings is discussed in detail in the instructional section of this manual. Here, when we talk about part programming we really mean . . . How do we input data so the computer can process this data and produce a part.

Data input is easily accomplished by typing in the information. The keyboard is arranged like an ordinary typewriter. When typing on the computer keyboard you **do not** make a carriage return, because it automatically goes to the next line when you exceed 32 characters on a line (spaces count too).

Merely typing data does nothing. You must press the enter key before the computer recognizes the entry and commits it to memory.

SPECIAL NOTE: When typing your part program statements into the computer be sure to line up the data in columns. It makes reviewing and editing programs much easier. The important thing to keep in mind when lining-up the X and Z statements is the decimal point locations. If you line-up the decimals and compensate for +/– signs it should look neat.

ONE CAUTION: Only one space is permitted between the G code and the block number. Therefore the G codes will not line up perfectly when you begin with block 100.

Your typed data should look like this on the screen:

```
10  'G92  X .300    Z .050
20  'G00  X–.075
30  'G01            Z–.800    F10
40  'G01  X .025
```

Inputting Data

Let's begin with a simple program. Remember every program statement (block) must be preceded by a block number. We suggest numbering by 10's so you will have room to insert additional data later if you need to. Remember to press enter after the last character on each line.

Type in this:

```
10  'G92  X .300    Z .050
20  'G00  X-.075
30  'G01            Z-.800    F10
40  'G01  X .025
50  'G00            Z .800
999  POKE 25,42: RUN
```

Did you make a mistake typing? If so don't worry. It's easy to correct. The key with an arrow pointing left (←) erases the last character. Press it more than once and it keeps moving back erasing one character at a time. If you've made a mistake on a line you already entered you can correct by typing the line over. You'll learn how to edit later.

Note: In this section we will not execute (run) any part programs. Actually running part programs is covered in the next section. You will save this program and use it then.

This program will result in a simple step being machined. The tool will advance into the material by .025 thus .050 of rough stock will be removed.

Before learning how to save the program you just wrote, check and make sure your program is exactly the same as the one listed in the manual.

To Verify it, do this:

1. Press the Clear key (small white key on extreme right of keyboard).

2. Type LIST
   This gives you a complete listing of the part program. You should see:

```
10  'G92  X .300    Z .050
20  'G00  X-.075
30  'G01            Z-.800    F10
40  'G01  X .025
50  'G00            Z .800
999  POKE 25,42: RUN
```

If your program is exactly like this then try saving it.

Saving a Part Program

Anytime you want a permanent copy of your part program you can save it on cassette tape. Saving a program is easy. Just follow these steps after you have typed a program into the computer memory.

1. Insert a blank cassette into the recorder. Be certain to manually advance the tape past the leader (if it has one), because the part program cannot be saved on the leader.

2. Press the recorder's play and record buttons at the same time until they lock down.

3. Type CSAVE "FILE NAME". The file name can be any sequence of eight or fewer characters or numbers. The file name must be enclosed in quotations. Use "your name-1" for the file name (i.e. JOHN-1).

4. Press enter.

When the program has been saved "OK" will appear on the screen. It should only take 3-4 seconds to save this short program.

Verifying the Saved Program

It is usually wise to verify that you have saved the program properly. Do this.

1. Rewind the tape
2. Press the play button until it locks down.
3. Type SKIPF "FILE NAME"

You should see the file name followed by "OK" if it is saved properly. If you get an "IO ERROR" then repeat the saving process with a fresh tape. Be sure to advance tape past the leader if it has one.

Note: We suggest buying a bulk tape eraser for erasing and reusing tapes.

Saving Many Programs On One Tape

You may save a number of part programs on one tape. The more separate part programs saved on one tape the longer the loading time required.

Always label your cassettes so you will know what the last part program on the tape is. By doing this it is easy to save additional part programs on the same tape.

To practice saving several programs on the same tape let's begin with the last program you just wrote. If you followed our instructions the file name should be "your name - 1" (ex. "JOHN - 1"). To load (even though it might already be there) do this.

1. Rewind the tape
2. Press play button recorder until it locks
3. Type: CLOAD "FILE NAME"
4. Press enter

The screen should show the "FILE NAME" and "OK" when the part program is loaded. It will only take a few seconds.

5. Type: LIST
6. Press enter

Your part program should be listed on the screen now. Let's add a couple of statements to this program.

Type in:

```
60  'G00   X-.050
70  'G01              Z-.500
```

Now type: LIST. Your screen should show:

```
10  'G92   X .300    Z .050
20  'G00   X-.075
30  'G01              Z-.800    F10
40  'G01   X .025
50  'G00              Z .800
60  'G00   X-.050
70  'G01              Z-.500
999   POKE 25,42: RUN
```

To save this variation of the program, do this:

1. Press the play and record buttons until they lock.
2. Type CSAVE "FILE NAME"
   Make the file name "your name-2" (ex. "JOHN-2")
3. Press enter

You should see "OK" on the screen beneath the file name. Verify the saved program by:

1. Rewind the tape.
2. Press the play button.
3. Type in SKIPF "FILE NAME" ("your name-2")

The file name and OK should show on the screen if saved properly.

Note: It will show the first file then skip past the one you designated.

Let's add the final three lines to this program. Type in:

```
80  'G01   X .025
90  'G00              Z .500
100   'G00  X .075
```

Type: LIST and your screen should show:

```
10  'G92   X .300     Z .050
20  'G00   X-.075
30  'G01              Z-.800     F10
40  'G01   X .025
50  'G00              Z .800
60  'G00   X-.050
70  'G01              Z-.500
80  'G01   X .025
90  'G00              Z .500
100  'G00  X .075
999  POKE 25,42: RUN
```

Save this just like the first two, but use "your name-3" for the file name.

You now have a cassette tape with three part programs saved. If you need to add another part program simply do this:

1. Rewind the tape
2. Press the play button
3. Type in SKIPF "FILE NAME" for the **last** part program. If you forgot what the last file name is press SKIPF and no "FILE NAME". Repeat typing SKIPF until you see the last file name, then redo with the correct "FILE NAME"
4. Press enter

The computer will skip through all three files and stop after the last file. Then you CSAVE a new "FILE NAME"

Listing Part Programs

Listing your program is quite easy. You know that because you've already done it several times. However, we listed the entire part program every time. Now you will learn how to list any part of the program you wish.

Let's just look at line 10.

Type:

LIST 10

Press enter

You should see this:

```
LIST 10
10  'G92   X .300     Z .050
```

Now let's look at lines 20 through 60.

Type:

LIST 20-60

Press enter

You should see this:

```
20  'G00   X-.075
30  'G01               Z-.800     F10
40  'G01   X .025
50  'G00               Z .800
60  'G00   X-.050
```

In order to see the last few lines of a program type in a block number close to the end and a dash (-). Try this:

LIST 90-

Press enter

You should see this:

```
90  'G00               Z .500
100  'G00  X .075
999  POKE 25,42: RUN
```

Remember to view any part of a part program all you need do is type LIST and the lines you wish to see.

3

Important Note

If you type LIST and garbage begins to scroll across the screen instead of your part program, then do this immediately

1. Press the red break key.
2. Press the clear key
3. Type in RUN and press enter.
4. Press break key again.
5. Type in LIST again and your part program should appear

Using the Editor

Before long you will desire to change an existing program, either to create a different part or to correct an error. In any case the editing capabilities of this computer allows changes to be made quickly.

Changing One Character

Let's say you made a mistake when you entered line (block) 70. The Z dimension should have been .300 **not** 500.

To edit line 70, all you do is:

1. Type EDIT 70
2. Press enter.

The screen should display all of line 70 **plus** 70 and a flashing cursor directly beneath the complete line 70. The flashing cursor is directly below the character that you can change.

To move the cursor into position for change simply press the space bar. The cursor will move one space at a time. To move backwards just press the backspace key (←)

Go ahead and position the cursor directly beneath the 5 by pressing the space bar.

To change the 5 to a 3 press C (for change) then type in a 3 and enter. PRESTO! the 5 is now a 3. Check it!

Type in LIST 70 and the line will read:

70 'G01 Z-.300

Changing More Than One Character

Let's say line 70 is still not correct. That Z dimension should really be .425 **not** .300. Type in:

1. EDIT 70
2. Press enter
3. Position the cursor beneath the first character to change

Determine how many characters are changing. In this case all three change.

4. Press 3 (3 characters are changing)
5. Press C (for change)
6. Type in 425 (the three new characters)
7. Press enter.

Of course, it's just as easy to change six or four or whatever.

Deleting Characters, Lines, Entire Programs

Let's say in line 40 the X dimension should have been .25 **not** 025. Getting rid of the zero is easy.

1. Type EDIT 40
2. Press enter
3. Position cursor beneath the 0
4. Press D (for delete)
5. Press enter

The zero is gone!

We will continue correcting line 40. Let's delete the entire X dimension (X .25).

1. Type EDIT 40
2. Press enter
3. Position cursor beneath X (first character to delete)
4. Press 5 (number of characters to delete – remember spaces count)
5. Press D (for delete)
6. Press enter

To delete an entire line just type in the line number and press enter

Try deleting line 90.

1. Type in 90
2. Press enter

This tells the computer that line 90 = 0. Press clear then type in LIST and press enter. See, line 90 is gone.

Deleting several lines or an entire program is also easy. We've changed quite a bit on this part program so let's look at how it is now. Do you remember how? That's right! Clear the screen, type in LIST and press enter.

Let's delete lines 30, 40 and 50. Here's what to do.

1. Type DEL 30-50
2. Press enter

That's all there is to it. Clear the screen and list your program and you will see lines 30, 40 and 50 are gone.

Deleting the entire program is really easy.

1. Type in DEL- (the dash means "to", so it means delete everything)
2. Press enter

Now if you list your program you will find nothing. We deleted the program from the computer memory but we still have it stored on tape.

You may also delete the entire program in the following manner.

1. Type in NEW
2. Press enter

Rewind your tape and reload "your name-3" so we can continue using the editor.

Inserting Characters

Line 40 is almost correct except .025 should be .0025. We need to insert a 0. You know by now how to begin.

1. Type EDIT 40
2. Press enter
3. Position the cursor beneath the 0
4. Press I (for insert)
5. Press 0
6. Press enter

Note: Once in the insertion mode you will remain in that mode until you press enter to conclude editing or shift ◆ at the same time to continue editing.

Extending a Line

Sometimes you may forget to enter a feed rate or a stop (M00) code It's no problem to correct that.

Let's say we want a feed rate in line 40 of 15 IPM.

1. EDIT 40
2. Press enter
3. Press X (for extend)
4. Space to line up beneath other feed rate (for neatness)
5. Type in F 15
6. Press enter

When you press X in the editing mode the cursor automatically goes to the end of that line and is in the insertion mode. *As you can see, editing is quick and painless, so don't worry about making mistakes, they are easy to correct.*

Common Errors

In most situations, the computer will tell you if you make a mistake. Listed below are error codes you may encounter frequently.

Error Code	Explanation
? SN ERROR	Syntax error. This usually means you misspelled a word (delt instead of edit) check your entry and type it in correctly. This error occurs frequently.
? IO ERROR	Input-Output error. If you experience this error while saving a program it is usually because you forgot to advance the leader on the tape. It also occurs when you attempt to record on top of another program. To correct you must repeat the loading or saving process.
? FC ERROR	Illegal function call error - This means you asked the computer to do something it can not do. This error will occur if you do not define the part dimensions properly for the graphics. Normally a check of the part diameter and part length data statements in the set-up will reveal the error. It also occurs if you omit a decimal point in an X or Z dimension.
BREAK IN 100	If you get this message as soon as you type in RUN it means the authorized electronic control cartridge has not been inserted in the computer. DO NOT INSERT WHILE COMPUTER IS ON.
? FM ERROR	BAD file mode - This error is ordinarily caused by accidently typing CLOAD instead of CLOADM when loading the master program.

USING THE COMPUTER
UNIT III
LEARNING ACTIVITIES

DIRECTIONS

The following activities are provided to allow you to reinforce your learning of Unit III. Complete each activity exactly as prescribed and do not proceed to Unit IV until all activities for Unit III are accomplished.

1. Look up each of the following words and phrases and then define them in your own words:

Ram Memory - _____

_____

_____

ROM Memory - _____

_____

_____

Microprocessor - _____

_____

_____

Bit - _____

_____

_____

Byte - _____

_____

_____

CLOAD - _____

_____

_____

CSAVE - _____

_____

_____

Enter - _____

_____

_____

Block Number - _____

_____

_____

Run - _____

_____

_____

Skipf - _____

_____

_____

_____

List - _____

_____

_____

Edit - _____

_____

_____

2. Complete all activities as identified in the background information section of this unit. Specifically this includes loading, saving, programming, entering data, listing, and editing.

USING THE COMPUTER
UNIT III
EXPECTED RESULTS OF LEARNING ACTIVITIES

1. RAM memory - Random access memory is the computer memory where programs are stored while the computer is turned on. When the computer is turned off all information in the RAM is lost.

   ROM Memory - Read Only Memory is the built in memory which stores programs the user does not manipulate. This could be considered permanent storage, and turning off the computer does not result in losing ROM.

   Microprocessor - The microprocessor is the central processing unit where the thinking is done in the computer.

   Bit - A binary notation that is a basic element of a computer which electronically carries the binary 1 or 0. Eight bits comprise one byte of information.

   Byte - A series of computer binary bits organized together to hold a symbol, letter, or number in a computer.

   CLOAD - A command given by the user, typed on the computer, telling the computer to start the cassette and place information on the tape in its RAM.

   CSAVE - A command given by the user, typed on the computer, telling the computer to start the cassette and record information in the computer onto this tape/cassette.

   Enter - Information written on the graphics screen must be followed by the typed word "enter", if the user wishes to place the information in memory.

   Block Number - Each block information in the program is assigned a block number from 1 - 998, usually in increments of 10.

   Run - A command typed into the computer, indicating that a program is to be executed.

   Skipf - When a program is being CLOADed, if more than one program is on a tape, the specific program can be gone directly to by specifying all programs to be skipped over prior to coming to the desired information/program.

   List - When a program is completed and the user wishes to review the program, he/she simply requests this by "LISTing".

   Edit - When changes, deletions, or additions are made through the computer it is termed edit.

2. The student should be able to load, save, program, enter, list and edit the computer.

USING THE COMPUTER
UNIT III
LAP EVALUATION

Name _____

A. Listed below are terms; phrases and statements frequently used in computer activities. Examine each statement closely and determine whether it is true or false. Indicate your answer on the line preceeding each.

1. _____ The master program is loaded into ROM.
2. _____ The Paxton-Patterson CNC unit is upgraded to 32K RAM.
3. _____ The main advantage to an upgraded RAM memory is that longer programs can be written.
4. _____ When loading the master program you type "CLOADMAS".
5. _____ Part programming generally begins by typing at the computer.
6. _____ When typing your part program statements into the computer, the data should be lined up in columns.
7. _____ The reason the block number is done by 10's is because it looks neater.
8. _____ When using a cassette the leader is included/used as part of the program.
9. _____ Only one part program can be placed on a tape.
10. _____ SKIPF is used when you are searching for a specific file on a cassette.
11. _____ When saving a program, only the record button should be pressed.
12. _____ Enter is only used when saving programs.
13. _____ CSAVE functions can generally be used for more than one part program per cassette.
14. _____ The list function can be used for a specific set of lines or an entire program.
15. _____ The edit function can be used on more than one line at a time.
16. _____ When pressing D in the edit mode, it means change.
17. _____ SN error means syntax error.
18. _____ Extending a line means taking away some piece of data.
19. _____ FM error means bad file mode and is generally caused by typing CLOAD rather than CLOADM.
20. _____ The electronic control cartridge which is designed to be inserted in the side of the computer should not be inserted while the computer is on.

B. Listed below are questions requiring short answers. Read each question carefully and then fill-in your answers on the blanks provided.

1. List and explain the basic steps for loading a program:

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

   d. _____

   _____

2. List and explain the basic steps for saving a program.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

   d. _____

   _____

3. List and explain the basic steps for listing a program:

   a. _____

   b. _____

4. List and explain the basic steps for changing two characters in a line:

   a. _____

   b. _____

   c. _____

   d. _____

   e. _____

   f. _____

   g. _____

USING THE COMPUTER
UNIT III
LAP EVALUATION KEY

A.
1. False
2. False
3. True
4. False
5. False
6. True
7. False
8. False
9. False
10. True
11. False
12. False
13. True
14. True
15. False
16. False
17. True
18. False
19. True
20. True

B.
1. a. Set volume control at 5 and load cassette into recorder.
   b. Press the play button on the recorder until it locks down.
   c. Type CLOADM for the master program (CLOAD for a part program)
   d. Press enter

2. a. Place a cassette into the recorder, being certain to advance beyond the leader or existing programs.
   b. Press the record and play buttons simultaneously, locking in position.
   c. Type CSAVE "File Name". File name can be any title but must be enclosed in quotations.
   d. Press enter.

3. a. Type; List
   b. Press enter

4. a. EDIT (type appropriate line number)
   b. Press enter
   c. Position the cursor beneath the first character to be changed
   d. Press 2 (2 characters are changing).
   e. Press C for change
   f. Type in the appropriate changes.
   g. Press enter.

UNIVERSITY OF
NORTH DAKOTA
DEPARTMENT OF INDUSTRIAL TECHNOLOGY

HIGH TECHNOLOGY MOBILE LABORATORY

Learning Activity Packet

MAJOR INSTRUCTIONAL UNIT: COMPUTER AIDED MANUFACTURING

LEARNING ACTIVITY PACKET (LAP) NUMBER: IV

LAP TITLE: CNC Part Programming

PREREQUISITE LAP (S) : III

LAP PURPOSE: The purpose of this LAP is to provide the student with information and activity pertaining to CNC part programming.

COMPETENCIES TO BE DEVELOPED: After completion of this unit the student should be familiar with basic information and activity related to CNC part programming.

LEARNING STEPS:

CNC PART PROGRAMMING

1. READ - Unit objective and specific objectives, pages 1 - 2.
2. STUDY - Background information, page 3
3. DO - Learning activities, pages 18 - 19.
4. STOP - See your instructor for evaluation. If your work is satisfactory, go to step 5. If not, go to step 2.
5. REVIEW - Objectives, background information, learning activities.
6. TAKE - The LAp evaluation, pages 22 - 23.
7. STOP - Give your instructor the completed LAP evaluation. If you scored 85 percent or more, go on to LAP V. If not, go to step 5.

CNC PART PROGRAMMING
UNIT IV
UNIT OBJECTIVE

After completion of this unit, the student should be familiar with basic information and activity related to CNC Part Programming. This familiarity will be evidenced by correctly performing the procedures outlined in the learning activities and by scoring at least 85 percent on the LAP unit evaluation.

SPECIFIC OBJECTIVES

After completing this unit, the student should be able to:

1. Identify and explain general information/activity commonly associated with CNC Part Programming, including processor languages, electronic institute association decimal point use, block number uses, feed rate, and miscellaneous function codes.
2. Identify and list preparatory function codes.
3. Identify and explain X and Z axis logic.
4. Identify and explain I and K center point dimension logic.
5. Identify and explain absolute and incremental programming.
6. Explain why circular interpolation is used in CNC work, and the necessary inputs.

CNC PART PROGRAMMING
UNIT IV
BACKGROUND INFORMATION

Manual Programming

In order for the student to fully understand the concepts of CNC machining it is necessary for them to perform manual programming. Manual programming simply means manually computing the cutting tool's path. The student should use a scientific calculator to assist in the mathematical computations.

No genuine understanding is possible without first mastering manual programming. The PAXTON/PATTERSON CNC TRAINING CENTER will allow the student opportunity to master this important topic.

NC Computer Processor Languages

While we will not attempt to teach these it is important to know what NC computer processor languages are. Basically, they are fast methods of computing complicated tool paths. It saves programmer's time in computing by using a program which does the computation with a minimum of data input.

Many NC computer processor languages exist today. Some of the more popular are COMPACT II, APT, ACTION, and ADAPT. They are useful because they save valuable programming time. For educational purposes it is better that the student fully understand the math involved in manual programming.

EIA RS 274D Standards

The Electronic Industries Association (EIA) is an important organization which has responsibility for developing standards for numerically controlled machine tools.

We have used EIA RS 274D so our product would be relevant. The function codes and axis motion designations are consistent with industrial CNC machine tools. EIA standards are used throughout the industrial community. The familiar preparatory function codes (i.e. G00, G01, etc.) are readily recognized by CNC part programmers and machine tool operators.

Decimal Point Programming

This system uses decimal point programming throughout. It is capable of interpreting two digits to the left of the decimal point and four to the right. It is unnecessary to use leading or trailing zeros. Of course, zeros between a number and a decimal point must be programmed.

EXAMPLES: 12.2456 4. 2.2 6.15 1.

Programming Format

In this section we will identify and explain the preparatory function codes and other data the student must know to write a part program. The explanations are in the order they appear on the PAXTON/PATTERSON CNC PROGRAMMING SHEET.

If the student understands this section he should have little difficulty using the CNC programming sheet for part program development.

Block Number - N

The first entry in a program statement is the block number. For the part program this can be any number from 1 to 998. Nearly one thousand blocks are available for part programming. Data cannot be entered without being preceded by a block number.

Block numbers are assigned for two reasons. 1) To indicate the sequence of information being processed. Thus, the order of machining steps and 2) to identify a particular statement which you may later wish to edit (i.e. it assists you in locating the data you wish to change).

We recommend assigning consecutive block numbers by increments of 10. (i.e. 10, 20, 30, etc.) This will allow an opportunity to add statements later if necessary.

IMPORTANT NOTE: Block 999 must always be as follows: 999 POKE 25, 42: RUN. Without this statement the program will not run.

Preparatory Function Codes - G

The preparatory function codes are very important because they set the mode of operation during actual machining. Each code is a two digit number preceded by G. All preparatory function codes are modal (i.e. unless changed the computer assumes you wish to continue with the same code), and only the apostrophe need be inserted. Each G code must be preceded by an apostrophe (').

All cutting moves (G01, G02, G03) are plotted on the monitor, but all G00 moves only show a + where they end. G00 moves actually erase any plotted line they contact.

Preparatory Function Codes

Code	Explanation
G00	Rapid traverse positioning move
G01	Straight line cutting move
G02	Clockwise circular cutting move
G03	Counterclockwise circular cutting move
G04	Establishes Dwell in seconds
G70	Establishes inch programming
G71	Establishes metric programming
G90	Establishes absolute programming mode
G91	Establishes incremental programming mode
G92	Establishes the initial start point. This defines the coordinate location of the tool at the start of the program. It is necessary in Absolute programming for proper machining, and in both Absolute and Incremental for correct graphics. A G92 code is required in **all** part programs.

Axis Definition - X & Z

The X & Z axes are reference lines established on all machines to standardize part programming. Lathe work utilizes two axes, the Z axis which is a line from the spindle center to the tailstock center, and the X axis which is perpendicular to the Z axis.

Dimensional data can be expressed by a maximum of 6 digits, two to the left of the decimal and four to the right (i.e. –12.3456, 1.05, –1.525, etc.). Decimal points should be programmed. Dimensional input should be preceded by the letter for the appropriate axis and the negative (–) algebraic sign if the tool should travel in a minus direction during incremental programming or to a negative location during absolute programming.

It is not necessary to use the positive (+) sign because the computer assumes the number is positive if no sign is used. Dimensional input actually describes coordinate locations on the appropriate axis when programming in absolute

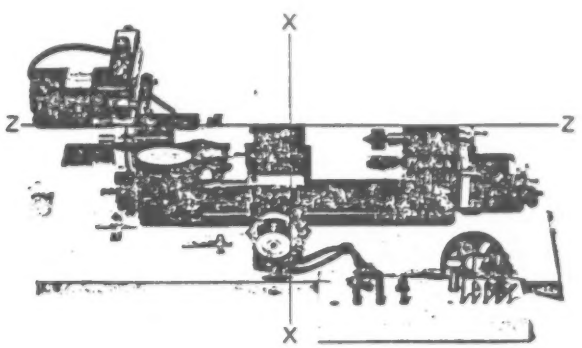


FIGURE 2-1
AXIS DEFINITION

When programming in the incremental mode, it is a negative direction on the Z axis when traveling toward the chuck and a positive direction when traveling toward the tailstock. It is a negative direction on the X axis when traveling toward the stock and positive direction when traveling away from the stock.

Note: When programming in absolute the sign depends on where the move ends in relation to the 0/0 point. See the section on absolute programming for further explanation.

Center Point Dimensions - I & K

Center point dimensions are programmed whenever an arc or partial arc is required. The I & K dimensions are in addition to the X & Z dimensions in a statement block.

These dimensions can be expressed as a six digit number (same as X & Z) preceded by I or K. Signs (+ –) are not required.

I – This describes the distance from the beginning of the arc to the centerpoint of the circle containing the radius along the X axis.

K – This dimension describes the distance from the beginning point of the arc to the centerpoint of the circle along the Z axis.

The I and/or K dimension must be computed whenever a G02 or G03 move is made. It usually requires some computation using right angle Trigonometry.

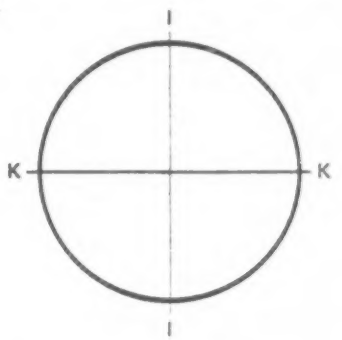


FIGURE 2-2
I & K AXIS
DEFINITION

If an arc begins on quadrant then only one I or K dimension is required. The other would be zero because it falls on the line. See the illustration below which shows the distance from the circle center point to the beginning of the arc.

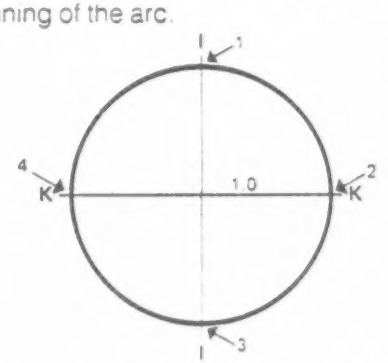


FIGURE 2-3
I & K DIMENSIONS
ON QUADRANT

1 = I 1 0 2 = K 1 0 3 = I 1 0 4 = K 1 0

Center Point Dimensions - I & K

Since each arc began on quadrant the distance from the center point was equal to the radius of the circle. No trigonometry is required in this situation.

Often times both I & K dimensions are required in the same block of information, because the partial arc does not begin on quadrant.

The next example shows a taper blending into a partial arc, thus requiring both I & K dimensions.

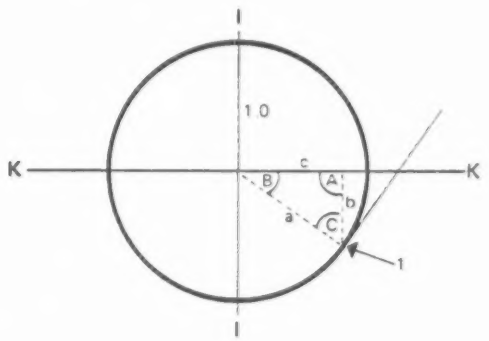


FIGURE 2-4
I & K DIMENSIONS
NOT ON QUADRANT

To solve for point #1 above it is necessary to construct a right triangle (dotted lines).

I = side b K = side c

The distance is actually measured from the I axis and K axis lines not the straight line distance from the center point to the starting point of the arc (side a). Otherwise the location would not be defined sufficiently, because the straight line distance would always be equal to the circle radius.

Assuming an angle on the taper of 35° (therefore C = 55°) and a one inch radius on the circle. The correct I & K dimensions are:

I = .5736 K = .8192

To program this would require a preparatory function code ('G03), a X dimension, a Z dimension and both the I & K dimensions.

Refer to Page 2-14 for explanation of Circular Interpolation.

Feed Rates - F

Feed rates are expressed in inches per minute (IPM). The maximum feed rate is 20 IPM, and is used automatically on all rapid traverse positioning (G00) moves.

Any feed rate from 1 to 20 can be used while turning. Feed rates are expressed by one or two digits preceded by an F (F10 = 10 IPM). These commands are modal, so they remain in effect until changed in a subsequent block. You may use a different feed rate for each block.

IMPORTANT NOTE: A feed rate **must** be used in every part program. It **must** be included in the first block where a cutting move occurs. If no feed rate is programmed the stepper motors will malfunction.

Miscellaneous Function Codes - M

Miscellaneous function codes perform various functions. Full sized machines use several M codes to control functions such as turning the spindle on and off or turning coolant on and off. We use only:

M00 Program stop, this is useful if you wish to stop the machining process to verify a dimension on the part or change a tool.

Remarks Section

The remarks section is for the programmer's benefit. It should summarize what the programmer expects to happen during that block. An arrow should indicate the direction of tool travel.

Example:

N	G	X	Z	I	K	F	M	REMARKS
10	G92	X .300	Z .050					ESTABLISH INITIAL START POINT
20	G00	X-.075						↑ POSITION FOR 1ST CUT
30	G01		Z-.800			F10		← MACHINE 1ST STEP

The remarks should be consistent with the dimensional data previously input in that block.

Absolute (G90) & Incremental (G91) Input

All part programs are developed by listing coordinate locations in the order necessary to generate the required part contour. There are two methods for defining coordinate locations. Absolute and Incremental.

The PAXTON/PATTERSON CNC TRAINING CENTER is capable of accepting either absolute or incremental, because the majority of controls in industry use both.

The codes G90 for absolute programming and G91 for incremental programming are modal and will remain in effect until cancelled by each other. No other G code will cancel these. The G90 or G91 statement should be in a block of information without any coordinate locations (axis dimensions). Unless changed to absolute the master program is in the Incremental mode.

Absolute Programming (G90)

When programming in the absolute mode all positions are expressed as dimensions from a single established zero point. The zero point will be the centerline and the faced stock. See illustration below.

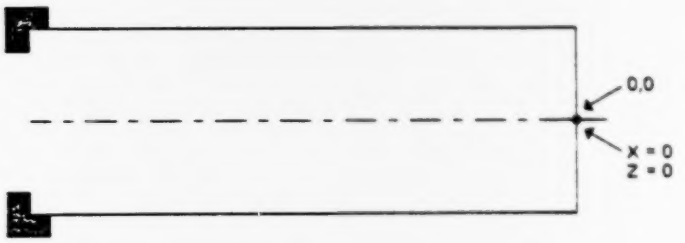


FIGURE 2-5
ZERO POINT FOR
ABSOLUTE PROGRAMMING

The illustration below shows how to sign (plus or minus) X and Z moves when programming in absolute. All X moves above the centerline are negative and those below are positive. All Z moves to the left of the zero point are negative and those to the right are positive.

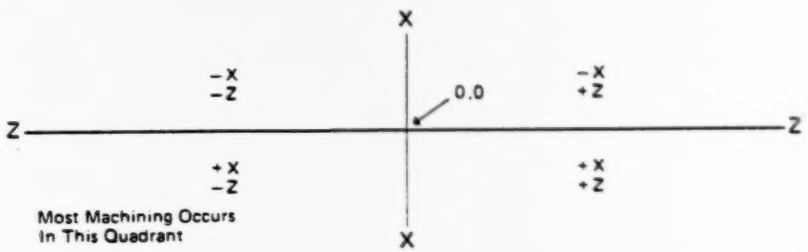
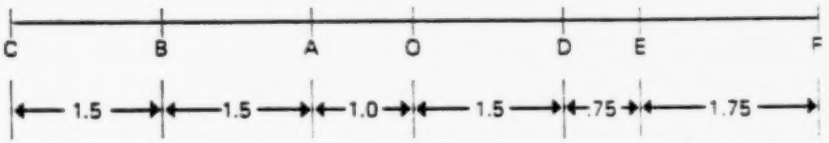


FIGURE 2-6
ABSOLUTE AXIS
DEFINITION

Absolute Dimensioning

Below is a diagram which illustrates how to express dimensions from point to point along the Z axis. Remember all dimensions are referenced from the zero point NOT the present point.



POSITION FROM	TO	COMMAND
O	A	Z – 1.0
A	B	Z – 2.5
B	C	Z – 4.0
C	D	Z 1.5
D	E	Z 2.25
E	F	Z 4.0
C	F	Z 4.0
C	O	Z 0

FIGURE 2-7
POSITIONS DESCRIBED
IN ABSOLUTE DIMENSIONS

The same method holds true on the X axis. Please note the algebraic signs (+/–) in the above illustration **do not indicate the direction of travel,** instead they denote the position relative to the zero point.

Start Point Dimensions

The start point dimensions define where the initial starting point for the tool is. The programmer must define exactly where the tool is positioned when the program begins.

The illustration below shows where the start point is and how to define it.

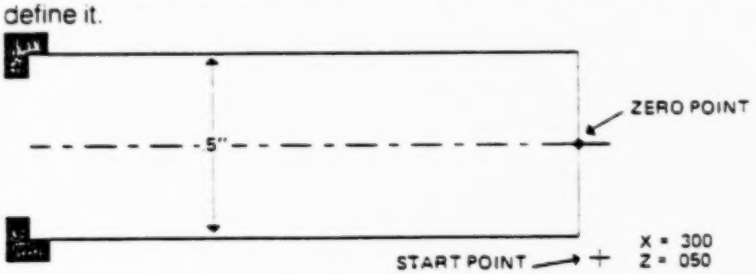


FIGURE 2-8
START POINT
DEFINITION

This illustration defines the start point as X = 300 and Z = 050, or .050 off the corner of the stock on both X & Z axes.

Start Point Code (G92)

The G92 pre-set register command is used to establish the X & Z axis control registers to the start point established by the programmer.

The program statement to pre-set the registers based on the previous illustration is: G92 X .300 Z .050
Without this statement the computer would not know where the tool was starting from when machining in the Absolute programming mode.

This program statement should be at the beginning of all programs.

Incremental Programming (G91)

All incremental program locations are given in terms of distance and direction from the immediately preceding point. In Incremental programming **the sign (+/–) indicates the direction of travel on each axis.**

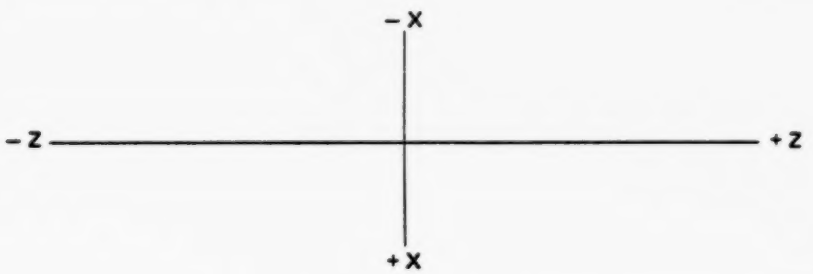


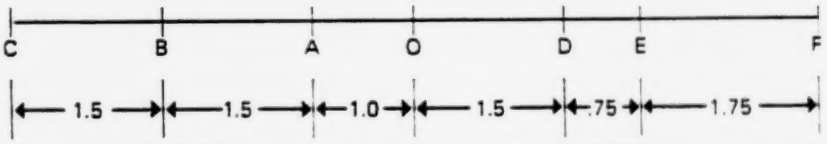
FIGURE 2-9
INCREMENTAL AXIS
DEFINITION

As the tool travels toward the headstock it is moving in a negative Z direction (left), and when traveling toward the tail stock it is moving in a positive Z direction (right). As the tool travels toward the material it is moving in a negative X direction, and when traveling away from the material it is moving in a positive X direction.

In incremental programming the signs (+/–) indicate the direction of tool travel.

Incremental Dimensioning

Below is a diagram which illustrates how to express dimensions from point to point along the Z axis. Refer to the similar diagram under absolute dimensioning for a comparison of the two methods



POSITION FROM	TO	COMMAND
O	A	Z – 1.0
A	B	Z – 1.5
B	C	Z – 1.5
C	D	Z 5.5
D	E	Z .75
E	F	Z 1.75
C	F	Z 8.0
C	O	Z 4.0

FIGURE 2-10
POSITIONS DESCRIBED
IN INCREMENTAL DIMENSIONS

Remember incremental dimensions are expressed from point to point, etc.

Start Point Code (G92)

As with absolute you must establish a start point so the graphics will begin at the right spot. To make this as easy as possible we use the G92 code in both absolute and incremental programming to set the start point. In incremental programs the 0/0 point is always the centerline and part face.

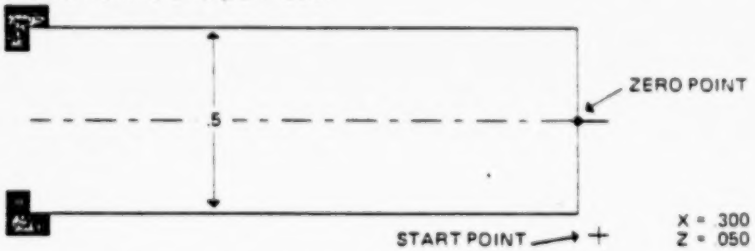


FIGURE 2-11
START POINT
DEFINITION IN
INCREMENTAL MODE

In this illustration the starting point was established .050 off the corner of the stock on both X & Z axis. Therefore the first program block should be

```
'G92     X .300     Z .050
```

An easy way to understand the difference between incremental and absolute is to think about giving directions. Ordinarily directions are given incrementally . . . go two blocks North, turn left and go two blocks West.

Linear Interpolation

Linear interpolation is a series of programmed points connected by straight lines. All machined shapes are the result of a series of straight line machining moves programmed in sufficient quantity to form an acceptable shape. In other words a circle is not really a continuous line, it is instead a large quantity of very short straight lines.

Straight-line moves (G01) use an accelerated linear interpolation method. Each line segment is joined end to end, but you are not required to program each individual line segment's beginning and end point. Without this accelerated process a simple part program could become unreasonably long.

Circular Interpolation

Circular interpolation greatly eases the time consuming programming necessary to generate arcs or partial arcs. To create a circular move it is necessary to program the following

1. The **Beginning Point of the Arc** (expressed in X and Z). This is normally the program block immediately preceding the GO2 or GO3 statement, since that positions the tool to the start point of the arc.
2. The **Direction of Travel (GO2 or GO3).** A GO2 move is clockwise and a GO3 move is counterclockwise.
3. The **End Point of the Arc** (expressed in X and Z). This will require both the X and a Z coordinate location.
4. The **Distance from the Circle Centerpoint to the Start of the Arc** (expressed in I and or K). Determining how to compute this is discussed on pages 2-6 and 2-7, Center Point Dimensions.

The circular interpolator breaks up the circular span into the smallest straight line resolution of movement which can be interpreted by the computer and the stepper motors. It then automatically computes enough of them to describe the cut and then generates the controlling signals sent to the stepper motors.

Without circular interpolation, a precision part containing a circular cut could require more than a thousand blocks of information.

PAXTON/PATTERSON

CNC Training Center

PROGRAMMING SHEET

DATE LABEL

N	G	X	Z	I	K	F	M	REMARKS

PART DIAMETER	PART LENGTH	STOCK DIAMETER

TYPE OF MATERIAL

PLEASE FEEL FREE TO DUPLICATE THIS FORM.

CNC PART PROGRAMMING
UNIT IV
LEARNING ACTIVITIES
DIRECTIONS

The following activities are provided to allow you to reinforce your learning of unit IV. Complete each activity exactly as prescribed and do not proceed to Unit V until all activities for Unit IV have been accomplished.

1. List the ten preparatory function codes, their explanations as identified in this unit, and the way each will be used.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

   d. _____

   _____

   e. _____

   _____

   f. _____

   _____

   g. _____

   _____

   h. _____

   _____

   i. _____

   _____

   j. _____

   _____

2. Observe the lathe and the X and Z axis. Draw a lathe diagram and identify the X and Z axes on your diagram, indicating why these axes are necessary.

   _____

   _____

   _____

   _____

3. Study figure 2-7 on page 2-10, and figure 2-10 on page 2-12. After studying, cover the command listings on each (do each one seperately) and have the student fill in the appropriate responses. Explain the differences in the two listings.

Absolute	Incremental
a.	a.
b.	b.
c.	c.
d.	d.
e.	e.
f.	f.
g.	g.
h.	h.

4. Study figure 2-8 on page 2-10. Draw a diagram of start point logic, identifying all information. Explain why this logic is necessary.

5. Complete activities one and two in the Paxton-Patterson CNC manual. Be sure to write all functions on the programming sheet.

CNC PART PROGRAMMING
UNIT IV
EXPECTED RESULTS OF LEARNING ACTIVITIES

1. a. G00 - Rapid Traverse Positioning Move. This will move the machine tool at maximum feed rate (F20).

   b. G01 - Straight line cutting move. Either the X or Z axis will move straight ahead.

   c. G02 - Clockwise circular cutting move. The cutting action will occur in a clockwise circular motion.

   d. G03 - Counterclockwise circular cutting. The cutting action will occur in a counterclockwise circular motion.

   e. G04 - Establishes dwell in seconds. This will cause a command to occur for a specified length of seconds.

   f. G70 - Establishes inch program. This connotes the english system is being used.

   g. G71 - Establishes metric programming. This connotes the metric system is being used.

   h. G90 - Establishes absolute programming mode. This connotes that the part programming is occuring in the absolute mode.

   i. G91 - Establishes incremental programming mode. This connotes that the part programming is occuring in the incremental mode.

   j. G92 - Establishes the initial start point. This is required in all programming to provide the start point, or reference point.

2.

The X and Z axes are necessary because this provides the basis upon which the conventional ways and crossfeed mechanisms can be identified and logically discussed in computer terms.

3.

Absolute	Incremental
a. -1.0	a. - 1.0
b. -2.5	b. -1.5
c. -4.0	c. -1.5
d. 1.5	d. 5.5
e. 2.25	e. .75
f. 4.0	f. 1.75
g. 4.0	g. 8.0
h. 0	h. 4.0

Absolute - All dimensions are expressed in terms of their relationship from the zero point where the X and Z axes cross, also identified as the 0,0 point.

Incremental - This is also sometimes called point to point, since after a given function is completed, the program dimension is specified from the most recent function.

4.

The start point is necessary as a consistent reference point for all work. It provides a basis for completing part programming in a similar manner time after time.

5. Activities one and two should help introduce the student to the machinery and its logic. The student will get a simple introduction to the programming sheet, start point positioning, and actually facing the brass workpiece. Be sure to have your instructor check your progress periodically.

CNC PART PROGRAMMING
UNIT IV
LAP EVALUATION

Name _____

A. Listed bewlow are terms, phrases and statements frequently used in CNC part programming. Examine each statement closely and determine whether it is true or false. Indicate your answer on the line preceeding each.

1. _____ Basic is a common NC computer processor language.
2. _____ The electronic industries association is primarily concerned with checking the electronic components in computers.
3. _____ Decimal point logic is used with the computer/CNC package since it is more precise and can easily be added on to one another.
4. _____ Nearly one thousand blocks are available for "block numbers" in programming.
5. _____ Preparatory function codes actually set the mode of operation during machining.
6. _____ Use of the G92 preparatory code is only necessary in absolute programs.
7. _____ The X axis runs at a 90^{o} angle to the ways.
8. _____ The Z axis runs parallel to the crossfeed.
9. _____ Center point dimensions are programmed whenever an arc or partial arc is required.
10. _____ Center point dimensions are generally expressed in Y and Q terms.
11. _____ The fastest feed rate, F10, is also the rapid traverse speed.
12. _____ Absolute programming is also sometimes called point to point.
13. _____ Incremental programming is based on start point logic.
14. _____ The zero point for absolute programming is X=0 and Y=0.
15. _____ The start point on our machine, when machining a piece .500 inch diameter, is Z=.050 and X=.300.
16. _____ Linear interpolation is a series of programmed points connected by straight lines.
17. _____ Circular interpolation requires additional time to complete a given machining exercise.

B. Listed below are questions requiring short answers. Read each question carefully and then fill-in your answers on the blanks provided.

1. Explain what I and K center point logic means:

   I = _____

   _____

   K = _____

   _____

2. Explain the basic differences between X and Z axes logic. _____

   _____

   _____

   _____

   _____

3. Explain the basic differences between absolute and incremental programming.

   _____

   _____

   _____

   _____

4. Explain why circular interpolation is used in CNC work. _____

   _____

   _____

   _____

   _____

5. List the necessary inputs for accomplishing circular interpolation.

   a. _____

   b. _____

   c. _____

   d. _____

CNC PART PROGRAMMING
UNIT IV
LAP EVALUATION KEY

A.
1. False
2. False
3. True
4. True
5. True
6. False
7. True
8. False
9. True
10. False
11. False
12. False
13. True
14. True
15. True
16. True
17. False

B.
1. I = The from the beginning of the arc to the centerpoint of the circle containing the radius along the X axis. K = The distance from the beginning point of the arc to the centerpoint of the circle along the Z axis.
2. X = The crossfeed motions and Z = The Way motions
3. Absolute programming says that all dimensions are calculated from the zero point where X and Z cross, while incremental is a point to point logic based on each function rather than the zero point.
4. Circular interpolation is used since it greatly eases the time consuming programming necessary to generate arcs or partial arcs.
5. The necessary inputs for accomplishing circular interpolation are:
   a. Beginning point of the arc
   b. Direction of travel
   c. End point of the arc
   d. Distance from the circle centerpoint Z. The start of the arc.

UNIVERSITY OF
NORTH DAKOTA
DEPARTMENT OF INDUSTRIAL TECHNOLOGY

HIGH TECHNOLOGY MOBILE LABORATORY

Learning Activity Packet

MAJOR INSTRUCTIONAL UNIT: COMPUTER AIDED MANUFACTURING

LEARNING ACTIVITY PACKET (LAP) NUMBER: V

LAP TITLE: Executing a Part Program

PREREQUISITE LAP (S) : III and IV

LAP PURPOSE: To provide the student with information and activity relative to part programming.

COMPETENCIES TO BE DEVELOPED:

After completion of this unit, the student should be able to set-up the lathe and computer, execute the program, a machined component, and troubleshoot the system.

LEARNING STEPS:

EXECUTING A PART PROGRAM

1. READ - Unit objective and specific objectives, page 3.
2. STUDY - Background information, pages 4 - 15.
3. DO - Learning activities, pages 16 - 17.
4. STOP - See your instructor for evaluation. If your work is satisfactory, go to step 5. If not, go to step 2.
5. REVIEW - Objectives, background information, learning activities.
6. TAKE - LAP evaluation, page 19.
7. STOP - Give your instructor the completed LAP evaluation. If you scored 85 percent or more go on LAP IV. If not, go to step 5.

EXECUTING A PART PROGRAM
UNIT V

UNIT OBJECTIVE

After completion of this unit, the student should be able to set-up and execute a part program. This knowledge will be evidenced by correctly performing the procedures outlined in the learning activities and by scoring at least 85 percent on the unit LAP evaluation.

SPECIFIC OBJECTIVES

After completing this unit, the student should be able to:

1. Set-up the lathe and computer for machining a component program.
2. Execute the program/component
3. View the graphics which are generated by the system.
4. Troubleshoot the system.

EXECUTING A PART PROGRAM
UNIT V
BACKGROUND INFORMATION

Executing a Part Program

Once a part program is loaded into the RAM memory of the computer (from cassette or by typing) the program can be executed or run.

To make understanding as easy as possible use the simple part program you developed in the last chapter.

First load the master program by following the instructions from the last chapter. Then load the program you developed.

Reminder on loading a part program.

1) Insert tape and rewind (if necessary)

2) Press play button on cassette player

3) Type CLOAD "FILE NAME"

4) Press enter

When the part program is found it will display the "FILE NAME" on the screen. When OK appears on the screen the program is loaded properly. To execute the program type in

RUN and press enter

You should see the prompt

SETUP

?

At this point the rough stock should be inserted in the chuck, the cutting tool selected and positioned to the start point.

To accomplish this, chuck up securely a 2" piece of brass. Next make certain the right angle roughing tool is correctly placed in tool holder.

Positioning Tool to Start Point

Turn lathe spindle motor on to approximately 750 RPM (between ¼ and ½ turn of speed control). Manually turn the hand wheels to position the cutting tool so it scratches completely around the extreme right hand edge of the rough stock. This is called "touching off".

Now back the tool out .050 in the X axis and .050 in the Z axis. Turning the handwheels counter clockwise will accomplish this. The start point dimension may vary at the part programmers discretion.

Note: If part drawing is not 1 to 1, you must adjust the set-up position accordingly. If your drawing had a part diameter of 4" and you were turning .5" stock, the amount you back off would be reduced to .0063 (.5 ÷ 4 = .125 X .050 = .0063). The computer automatically factors all other dimensions within the part program.

Your lathe should now be properly set-up to begin machining. After the set-up is complete, type in OK, press enter and the screen should display:

PART DIA. = ?

Part Diameter

The computer needs to know what the rough stock diameter is **according to the drawing** not actual stock diameter in the chuck. This information is used to establish the size of the part produced on the video monitor.

The correct part diameter for your part is .5 (because this part was drawn to actual stock size).

Type in .5 and press enter.

Part Length

The next prompt displayed on the screen should be

PART LEN. = ?

You must establish how much the part will extend from the chuck according to the drawing. This establishes the second dimension for the Graphics Display.

The computer will automatically scale the size up or down to provide the maximum viewing area for the part. This means students can use full sized part drawings instead of being confined to the size of either the monitor screen or lathe capacity.

Again, for this part the part length is 1.0". The length of cut was .800", but it is best to add .20 - .25" to insure the tool does not strike the chuck. **Caution -** this will not prevent a crash if the stock is actually set up wrong. There must be at least 1" extending beyond the chuck jaws for this program to run safely.

Type in 1.0 and press enter

The last prompt appearing on the screen is:

STOCK DIA. ?

Stock Diameter

The stock diameter is the diameter of the rough stock you are about to machine. If you are using the brass stock which came with the CNC Training Center, the correct stock diameter would be .5 (½"). *DO NOT ENTER .5 NOW.*

The stock diameter dimension is extremely important because when a dimension is entered the electronic motor control cartridge begins sending electrical impulses to the stepper motors to produce axis motion.

If no dimensional data is input only the graphics appear; no lathe axis motion occurs. This is extremely useful, because it allows the part to be verified on the video monitor prior to actual machining.

Right now the following should be displayed on the screen:

RUN
SETUP
?
PART DIA. = ? .5
PART LEN. = ? 1.0
STOCK DIA. ?

Let's view the graphics only. Do not input any dimensional data for stock diameter.

Press enter

You should see the part being graphically reproduced on the screen. It should look like this:

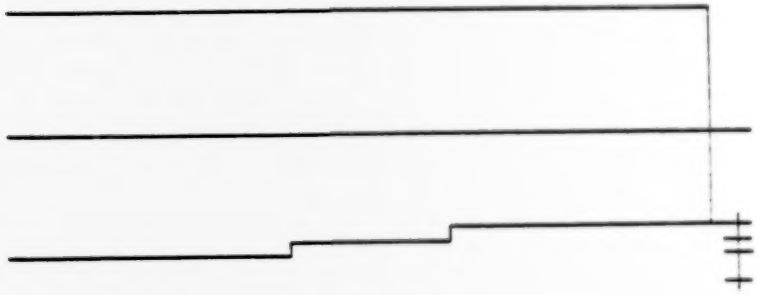


FIGURE 4-1
DEMONSTRATION PART–
GRAPHIC DISPLAY

Repeating the Graphics

If you would like to see the graphics again simply press the enter key. You should see the graphics disappear and textual data appear. The text will be the data you entered previously PLUS this important prompt:

NEW SCALE - Y OR N?

You now have the option of changing the scale (i.e. part diameter and part length). To change it:

1. Type in a Y for yes
2. Press the enter key

Now you can change the part diameter or part length by answering the prompts differently. Usually you would not want to change these if the graphics look correct. To continue using the same scale simply:

1. Press the enter key (it is not necessary to type in N for no the computer assumes no if you do not type in Y).
2. Press the enter key again when the prompt for stock diameter appears.

The graphics should begin again. Therefore, to view the graphics repeatedly just press the enter key three times.

As you can see, this is simply two steps which removed .100 (tool moved .025 each pass). If your screen looked like the illustration, let's try to machine this part. Verify your set-up of the lathe. Recheck that at least one inch of stock is extending from the chuck.

Your part should now be graphically illustrated on the monitor. In order to machine the part, do this:

1. Press the enter key and you should see

   NEW SCALE - Y OR N?

2. Press the enter key again if the scale was correct. You should see

   STOCK DIA?

3. Type in the actual rough stock diameter. (.5 if you are using the supplied brass) but DO NOT press the enter key yet.

Note: Stock diameter does not have to be identical to the part drawing diameter. The computer will automatically scale the part and machine to a proportionally smaller or larger part. However, you must manually scale your start point dimensions and the physical position of the cutting tool (see the Note under Positioning Tool To Start Point on Page 4-3).

Prior to pressing the enter key:

- LOWER LATHE GUARD AND PUT SAFETY GLASSES ON.
- Turn on the spindle with toggle switch and adjust spindle speed to approximately 750 RPM (between ¼ and ½ a turn of the speed control).
- Make sure the power supply is on.

After you complete the above steps press the enter key and you should observe the graphics and the lathe machining the part.

Programming Sheet

Here is what the programming sheet looks like for this part.

DATE: LABEL:

N	G	X	Z	I	K	F	M	REMARKS
10	G92	X .300	Z .050					ESTABLISH INITIAL START POINT
20	G00	X-.075						↑ POSITION FOR 1ST CUT
30	G01		Z-.800			F10		← MACHINE 1ST STEP
40	G01	X .025						↓ SQUARE OFF CUT
50	G00		Z .800					→ POSITION
60	G00	X-.050						↑ POSITION FOR 2ND CUT
70	G01		Z-.500					← MACHINE 2ND STEP
80	G01	X .025						↓ SQUARE OFF CUT
90	G00		Z .500					→ POSITION
100	G00	X .075						↓ POSITION TO START POINT
999	POKE 25,42: RUN							

Review

Remember, to execute a program after you type in or load from a cassette merely type in the command:

RUN

Then the computer will give four prompts. They are:

SET UP? Physically position the cutting tool to the start point. This will require touching off and backing off the tool to the programmed start point (G92).

PART DIA. = ? Enter the part diameter of the rough stock as it appears on the part drawing. It can be larger or smaller than the screen viewing area because the computer will automatically size it for optimum viewing.

PART LEN. = ? Enter the length of the part from the part drawing. Be sure to add at least .25" to the length so it will finish the cut away from the chuck jaws. You must also make sure the actual stock extends from the chuck this far (if scale is 1-to-1).

STOCK DIA.? Enter the actual rough stock diameter of the material in the chuck. This enables the computer to scale down the part if the stock being turned is smaller than the drawing dimensions.

If no stock diameter is entered, only the graphics will display. No machining will occur.

Note: It may be useful to include a block of information after your part program which gives all the set-up dimensions. This would be a remark statement.

Example:

```
990 REM  PART DIAMETER = .5
         PART LENGTH = 1.0
         STOCK DIAMETER = .5 (or actual if different)
```

By doing this you will not need to remember the set-up information.

PAXTON/PATTERSON
CNC Training Center

5

Helpful Hints

1. Always load the master program (CLOADM) at the beginning of the day or class.
2. Type in NEW before entering a new part program into the computer memory.
3. Use CLOAD to load part programs from cassette tape to the computer.
4. Press the Break (red) key twice if you want to edit or list a program after viewing the graphics.
5. To interrupt the machining process press the Reset button on the right rear of the conmputer. Be careful not to turn off the computer by pressing the button on the left rear . . . you will lose the memory.
6. To view the graphics again after you have seen it once, press the enter key three times.
7. Always advance cassette tapes past the leader before attempting to save a part program.
8. When saving a part program (CSAVE) be sure to press both the Play and Record buttons until they lock down.
9. Always verify a saved program by using SKIPF.
10. Do not attempt to turn the handwheels on the lathe after machining has begun.
11. The graphics on the monitor are always scaled up or down to provide the optimum viewing area. They will not be actual dimensions.
12. The maximum depth of cut should not exceed an actual .035" in brass.
13. Never insert the Electronic Motor Control Cartridge while the computer is on . . . serious damage could occur which is not covered by the warranty.
14. Be sure to use the negative (–) sign when appropriate.
15. The only time you will have a negative X move when programming in Absolute is when you face the part. Any other negative X move would cut the part in half because the tool would be beyond the centerline.
16. In Absolute programming all coordinate locations (X & Z) are expressed from the Zero point. All X & Z data indicate location **not** distance of travel.
17. When "touching off" be sure to lightly scratch the rough stock. If you scratch too deeply you could cause the finish diameter to be too small because the set-up would be inaccurate.

5

Trouble Shooting Guide

If you experience any problems with the Paxton/Patterson CNC Training Center check the following table for symptoms and possible cures.

If you can't correct the problem on your own, call your local Paxton/Patterson or Graves-Humphreys representative.

SYMPTOM	EXPLANATION OR CURE
Stepper motors do not move properly (they may move sporadically)	1. Interference from spindle motor. Plug into different outlet. Never plug into power strip. 2. Electronic Control Cartridge not inserted properly. Turn off computer and insert cartridge again. 3. Possible power line fluctuations. 4. Loose connections on power supply.
Program will not execute and indicates break in 100	Electronic Control Cartridge is not inserted.
Picture on video monitor breaks up or is not good	1. Video/Computer connections are loose. Tighten all connections. 2. Readjust contrast and black level on monitor.
Program executes in graphics but not on lathe	1. Power supply not turned on. 2. Lathe not connected to control cartridge.
Garbage scrolls across monitor when you type list	1. Immediately press break key (red) then clear key. 2. Type in RUN and press enter. 3. Press break again. 4. Type in LIST.

Trouble Shooting Guide

SYMPTOM	EXPLANATION OR CURE
Part program executes slowly on the monitor	You have probably accidently entered a stock diameter, which would slow the graphics to the atual machining rate. To correct, press the reset button on the right rear of the computer, then enter only the part diameter and part length.
The stepper motors are moving the tool in the opposite direction they should. Any totally erratic axis motion	This usually occurs because no feed rate was specified in the first block where a cutting move occurs. To correct, edit the first block where a cutting move occurs to include a feed rate of less than 20 IPM. Usually the first cutting move is G01.
The cutting tool does not move past the centerline, thus rough stock is left in the center after facing.	1. The X dimension was not programmed correctly to move past center. 2. The set-up of tool start point was incorrect. This can happen if the part diameter and stock diameter are different, and the operator does not factor the set-up dimensions (see Page 4-3)
Excessive tool noise or chatter.	This could be caused by– 1. The depth of cut is too large 2. The rough stock extending too far from the chuck without support 3. The tool bit not angled properly to provide relief 4. Feed rate set too fast 5. Spindle speed too fast 6. The material is too hard for the programmed speed and feed rate

Trouble Shooting Guide

SYMPTOM	EXPLANATION OR CURE
Tool carriage does not move easily or is binding	It is possible chips (particularly plastic chips) have clogged the lead screws and are retarding movement. To correct, clean the lead screws thoroughly and re-lubricate.
No output whatsoever to the video monitor	1. The cable is not attached to both the video monitor jack and the video monitor. 2. Video monitor cable is plugged into the TV jack on the computer instead of the special video monitor jack installed directly above the TV jack.
Stepper motors are not operating at all	1. Power supply is not turned on. 2. Electronic motor control cartridge is not inserted in computer 3. Cables from electronic motor control cartridge are not connected to stepper motors. 4. No stock diameter was entered
Axis movement is reversed (i.e. X moves when Z should)	1 The cables from the electronic motor control cartridge are connected backwards. Reverse the connections to the stepper motors to correct. 2. Programming error.
When you type in RUN to execute a part program and you get OK instead of SET-UP?	You probably forgot to add Statement 999 to your part program. If so, simply type in 999 POKE 25,42:RUN.

EXECUTING A PART PROGRAM
UNIT V
LEARNING ACTIVITIES

DIRECTIONS

The following activities are provided to allow you to reinforce your learning of Unit V. Complete each activity exactly as prescribed and do not proceed until all activities have been accomplished.

1. Explain what each of the following prompts means, in your own words:

   a. Set-up _____

   _____

   _____

   b. Part Dia. _____

   _____

   _____

   c. Part Len. _____

   _____

   _____

   d. Stock Dia. _____

   _____

   _____

2. Explain the basic steps necessary for positioning the cutter relative to the workpiece, for start point:

   a. _____

   _____

   b. _____

   _____

3. Explain the basic steps necessary for completing the machining of a component.

   a. _____

   _____

   b. _____

   _____

   c. _____

   _____

   d. _____

   _____

4. Complete all activities as outlined in the Unit V background information.
5. Complete as many activities from numbers 3 - 15 in the Paxton-Patterson workbook as time will permit.
6. Review all points covered under "trouble shooting" in this unit, as related to the actual machinery (i.e., actually check out the equipment).

EXECUTING A PART PROGRAM
UNIT V
EXPECTED RESULTS OF LEARNING ACTIVITIES

1. a. Set-up - This prompt requests that the operator make the appropriate adjustments in preparation for the operation.
   b. Part Dia. - This prompt requests that the operator provide part diameter according to the drawing specifications.
   c. Part Len. - This prompt requests that the operator provide part length according to the drawing specifications.
   d. Stock Dia. - This prompt requests that the operator provide actual stock diameter of the material about to be machined.

2. a. Touch off from the right edge of the workpiece nearest the operator.
   b. Back the tool out 0.050 inch in the X axis and over to the right 0.050 inch in the Z axis.

3. a. Set-up the machine/workpiece
   b. "Touch off" with the tool.
   c. Complete all prompts with motor started at appropriate RPM.
   d. Follow through on graphics and machining.

4. It is expected that the student will gain sufficient competencies to enable them to execute a part program.

5. It is expected that the student will gain competencies, gradually, as they work through all activities in the Paxton-Patterson CNC package.

6. It is expected that students will become competent at trouble shooting and problem solving potential problems in the CNC system.

EXECUTING A PART PROGRAM
UNIT V
LAP EVALUATION

Name _____

A. Listed below are terms, phrases and statements frequently used in understanding how to execute a part program. Examine each statement closely and determine whether it is true or false. Indicate your answer on the line preceeding each.

1. _____ After a part program is loaded into the RAM memory of the computer the program can be executed.
2. _____ The "set-up" prompt automatically indicates if the set-up is done properly.
3. _____ Positioning is properly done by randomly locating the tool in reference to the workpiece.
4. _____ "OK" is used only with the set-up prompt.
5. _____ Part diameter should be entered as the actual expected part diameter.
6. _____ The computer will automatically scale the dimensions to allow a component to fit on the screen, graphically.
7. _____ When the stock diameter prompt appears, it is automatically time to provide the dimension as soon as possible.
8. _____ The graphics of a part are viewed by pressing "G" and entering.
9. _____ After viewing part graphics it is necessary to provide stock length.
10. _____ Stepper motors moving sporadically could be due to interference from the spindle motor.
11. _____ A break in line 100 indicates that the steppers are not plugged into the motors.
12. _____ Chatter at the workpiece during cutting may be due to taking more than .035 inch depth.
13. _____ If the stepped motors are not being energized it may be due to not entering the stock diameter.

B. Listed below are questions requiring short answers. Read each question and then fill-in your answers on the blanks provided.

1. Explain two reasons why the graphics may be useful in the CNC system.

   a. _____

   _____

   b. _____

   _____

2. Explain four possible errors/circumstances which could help explain excessive tool noise or chatter.

   a. _____

   b. _____

   c. _____

   d. _____

EXECUTING A PART PROGRAM
UNIT V
LAP EXALUATION KEY

A.
1. True
2. False
3. False
4. True
5. True
6. True
7. False
8. False
9. False
10. True
11. False
12. True
13. True

B. 1. a. They permit the operator to "check" the program without using actual cutting.

b. The cutting path can be observed prior to actually cutting.

2. a. Too great a cutting depth.
b. Inadequate clamping/chucking
c. Incorrectly prepared cutting tools.
d. Too fast a feedrate.
e. RPM too fast.
f. Material to be machined may be too hard.